AASHTOWare BrD/BrR 6.8

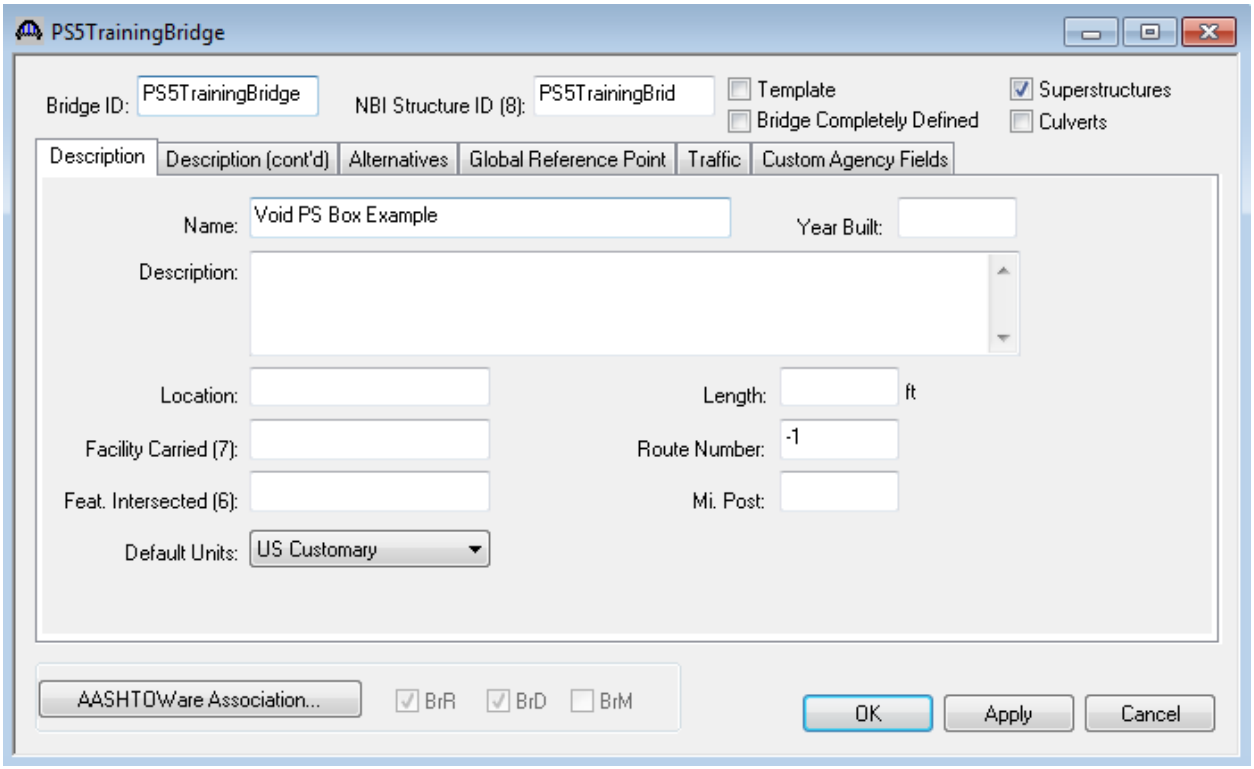
Prestressed Concrete Structure Tutorial

PS5 – Void Prestressed Box Beam Example

BrR and BrD Training

PS5 – Void Prestressed Box Beam Example

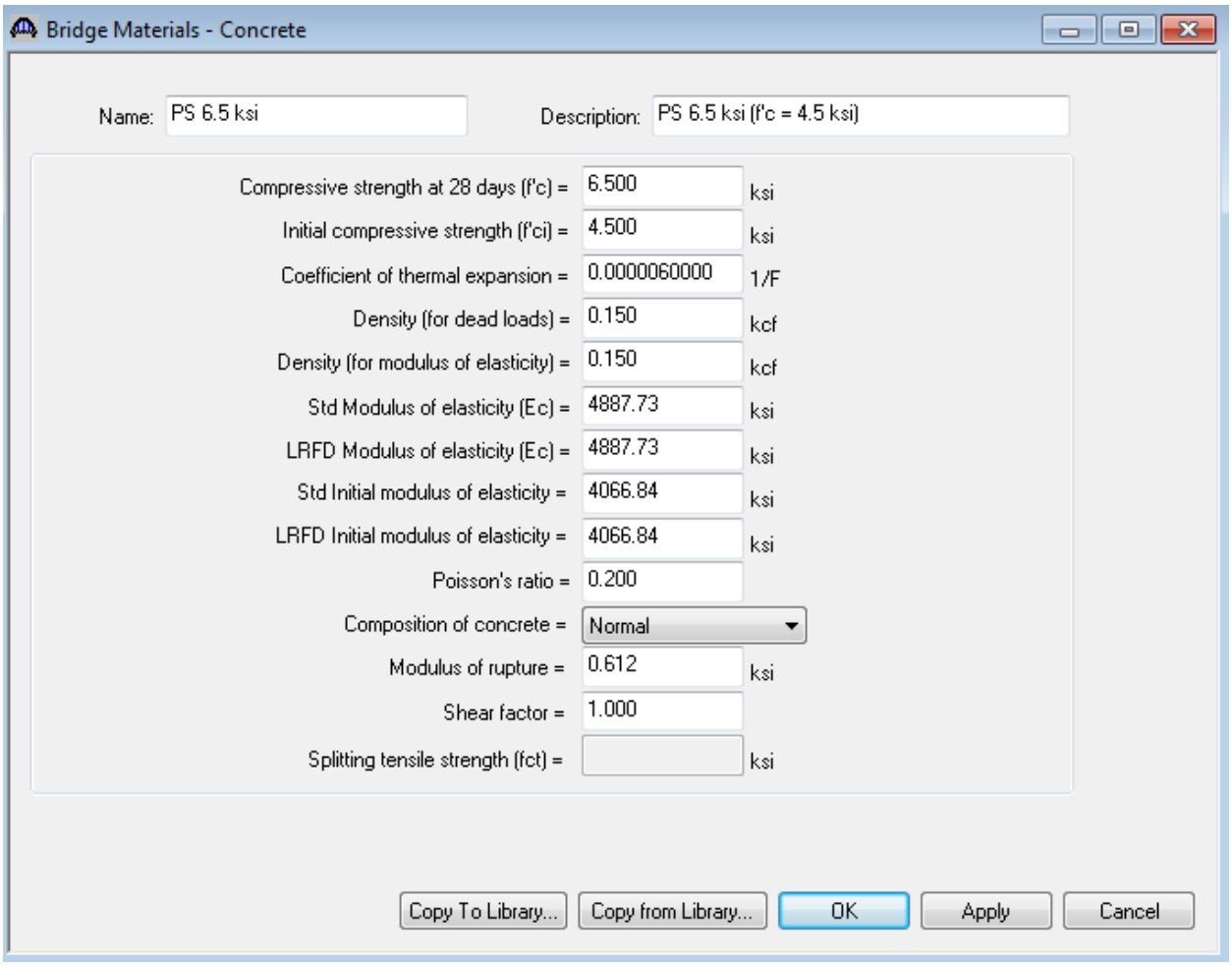
From the Bridge Explorer create a new bridge and enter the following description data:



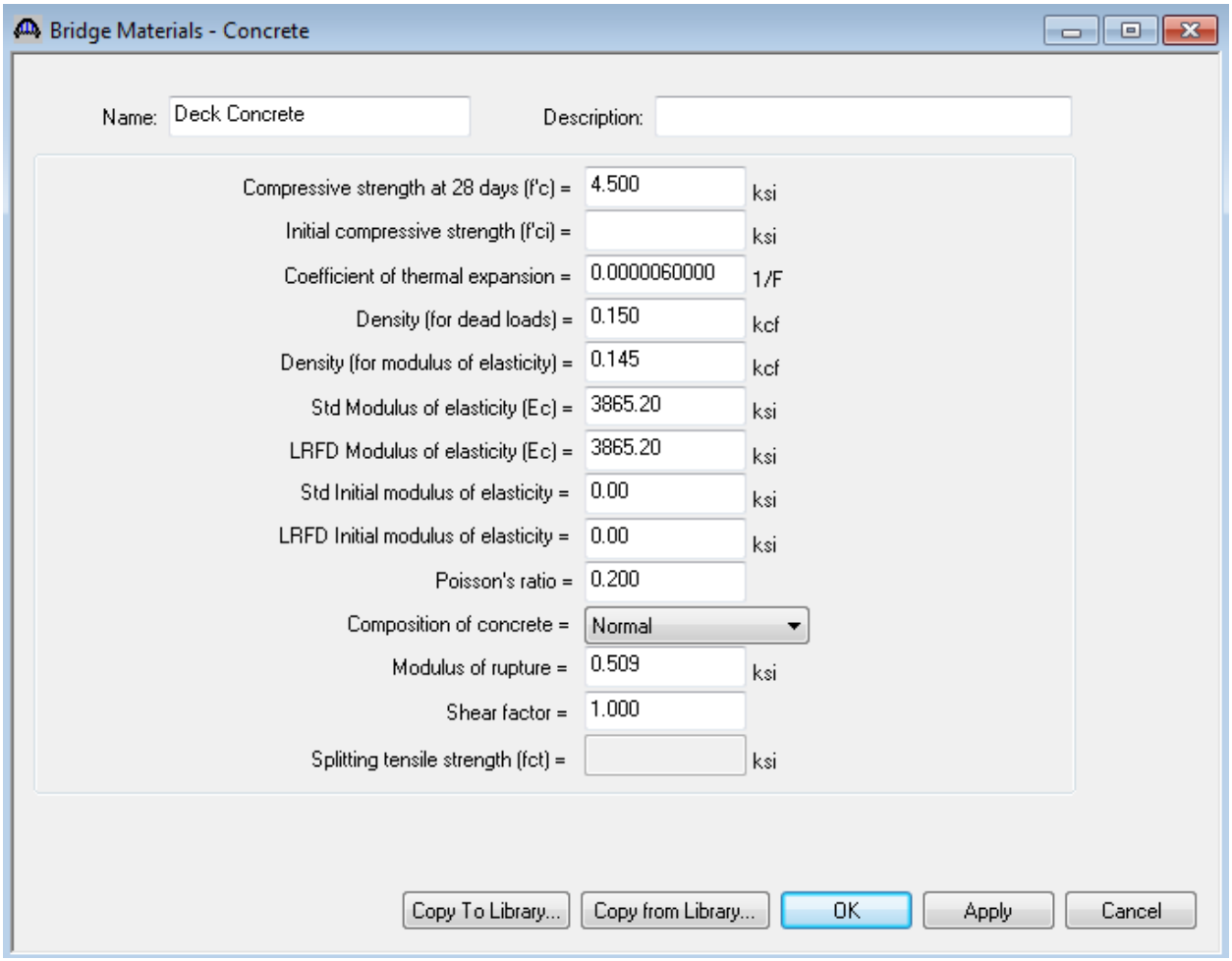
Close the window by clicking Ok. This saves the data to memory and closes the window.

To enter the materials to be used by members of the bridge, click on the ⊞ to expand the tree for Materials. To add a new concrete material click on Concrete in the tree and select File/New from the menu (or right mouse click on Concrete and select New).

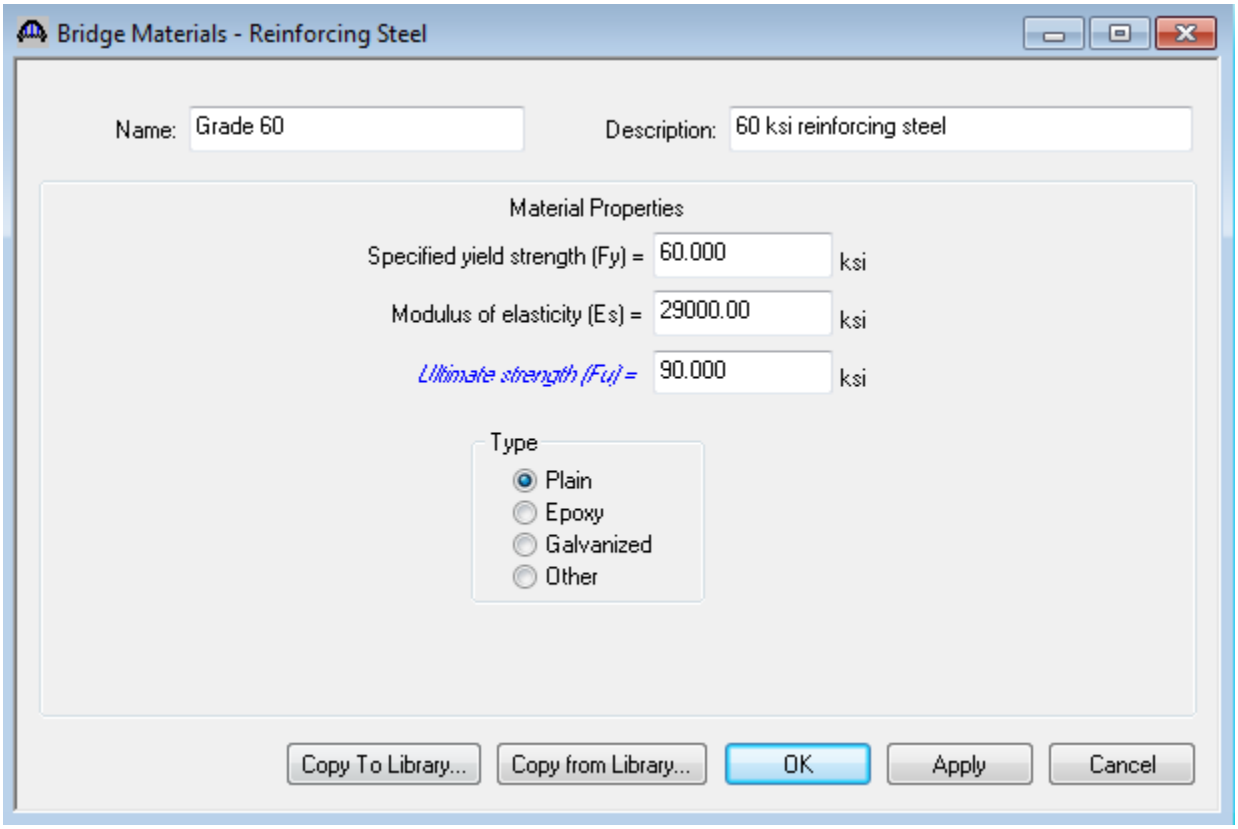
Add the concrete material "PS 6.5 ksi" by entering the data as follows. Click Ok to save the data to memory and close the window. This concrete will be used for the beam concrete in this example.



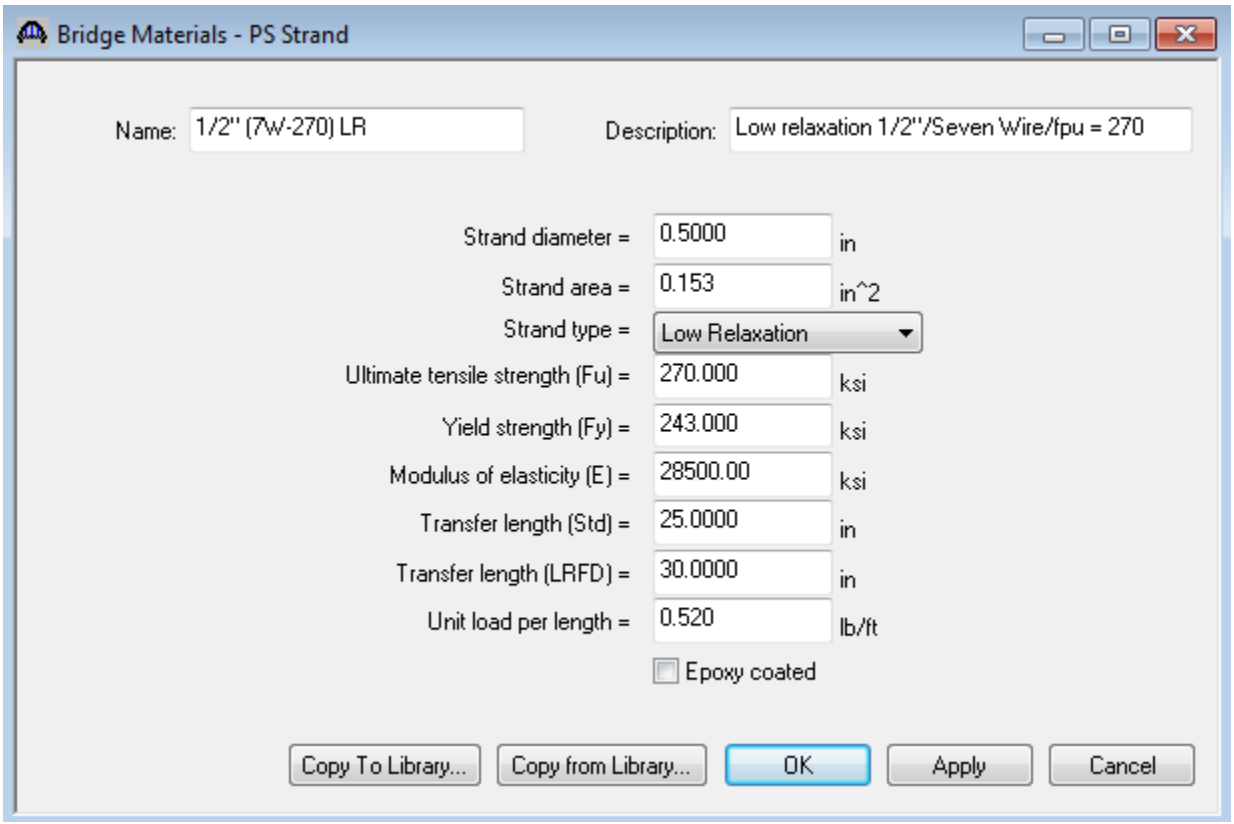
Add a concrete material for the deck by entering the following data. The window will look like the one shown below:



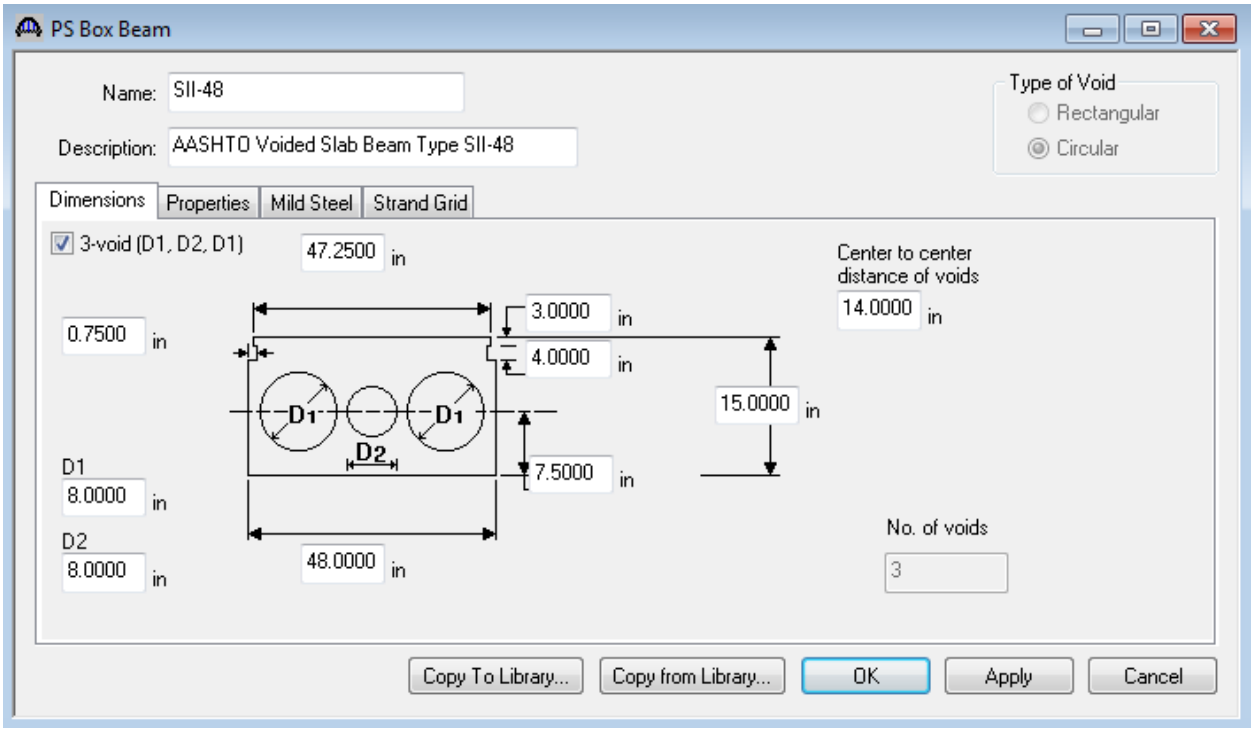
Add a reinforcement material using the Copy from Library techniques. The windows will look like the one shown below:



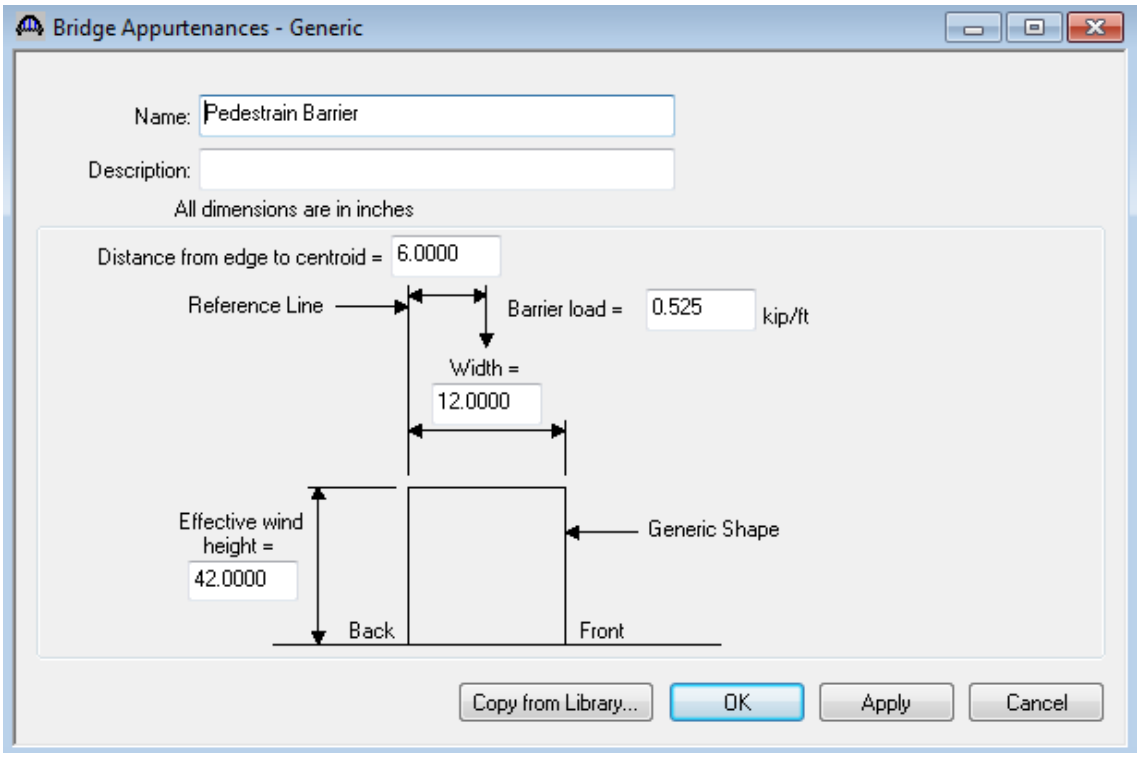
Add a prestress strand using the Copy from Library techniques. The windows will look like the one shown below:



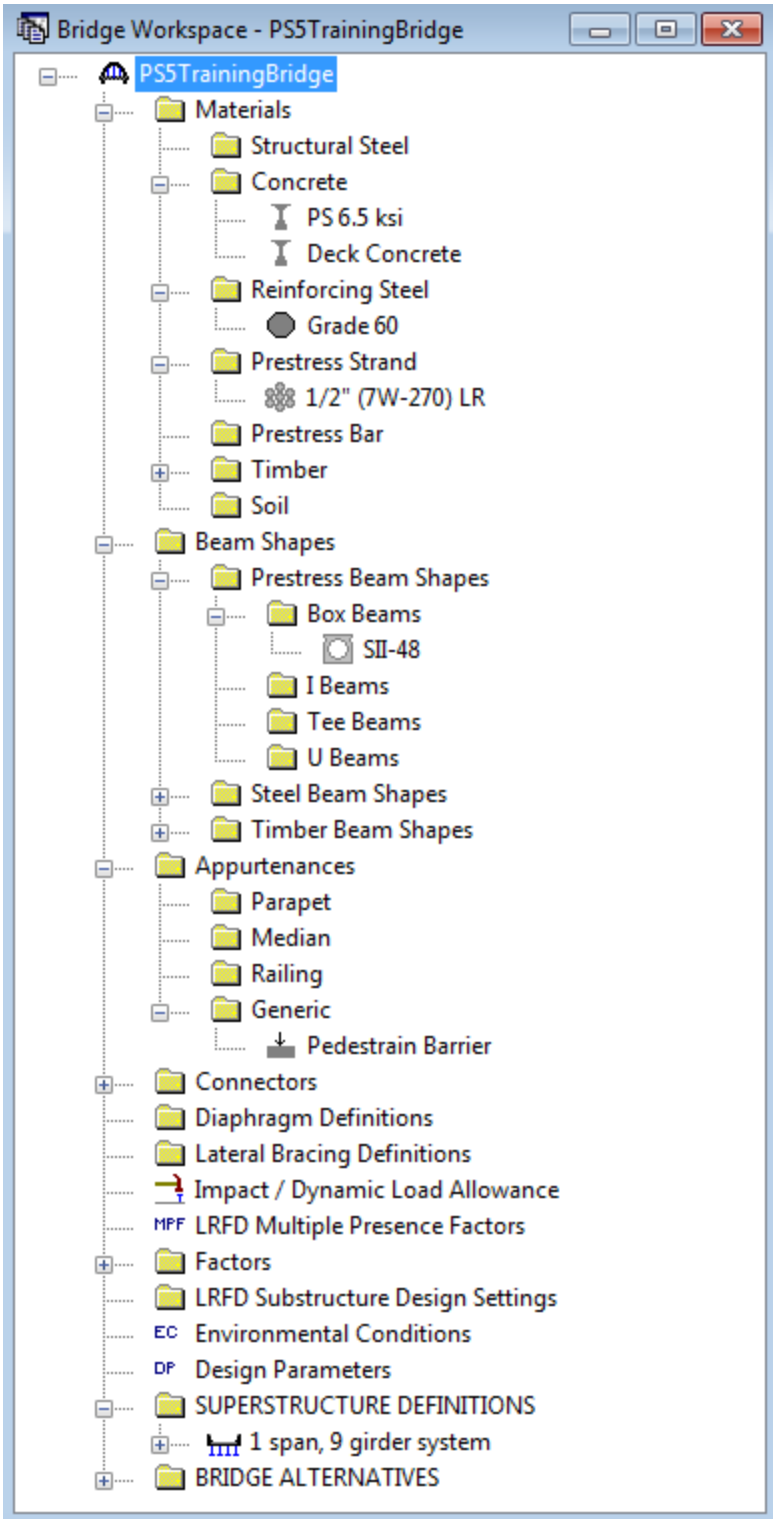
To enter a prestress beam shape to be used in this bridge expand the tree item labeled Beam Shapes. Click on Box Beams in the tree and select File/New from the menu (or double click on Box Beams in the tree). Select the Void Type as Circular and click on the Copy from Library button. Select SII-48 and click Ok. The beam properties are copied to the Box Beam window as shown below.



To enter the appurtenances to be used within the bridge expand the tree branch labeled Appurtenances. This bridge has a concrete pedestrian railing. Define a Generic appurtenance to model the concrete railing. The completed window is shown below:

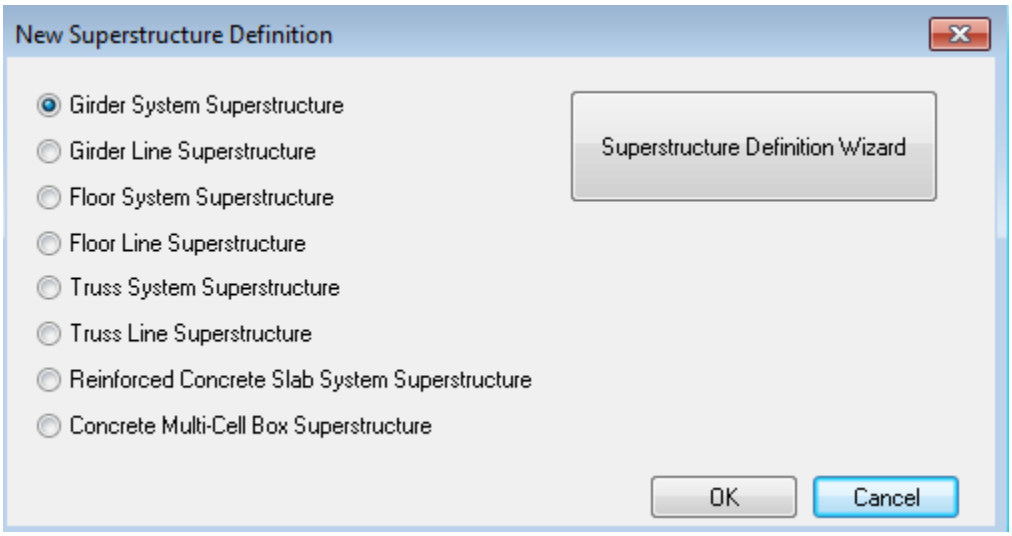


The Bridge Workspace is shown below:

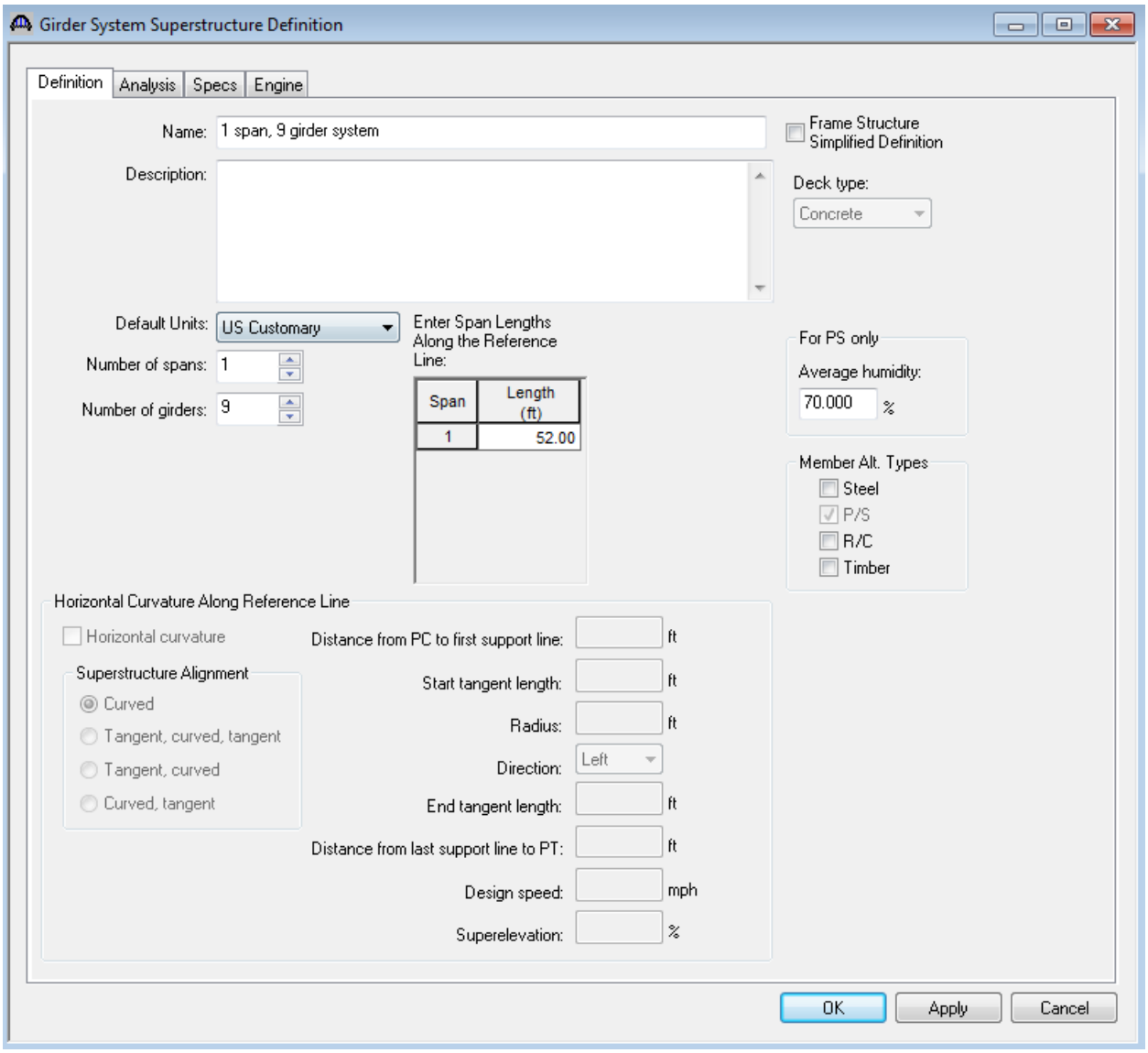


The default impact factors, standard LRFD and LFD factors will be used so we will skip to Structure Definition. Bridge Alternatives will be added after we enter the Structure Definition.

Double click on SUPERSTRUCTURE DEFINITIONS (or click on SUPERSTRUCTURE DEFINITIONS and select File/New from the menu or right mouse click on SUPERSTRUCTURE DEFINITIONS and select New from the popup menu) to create a new structure definition. The dialog shown below will appear.

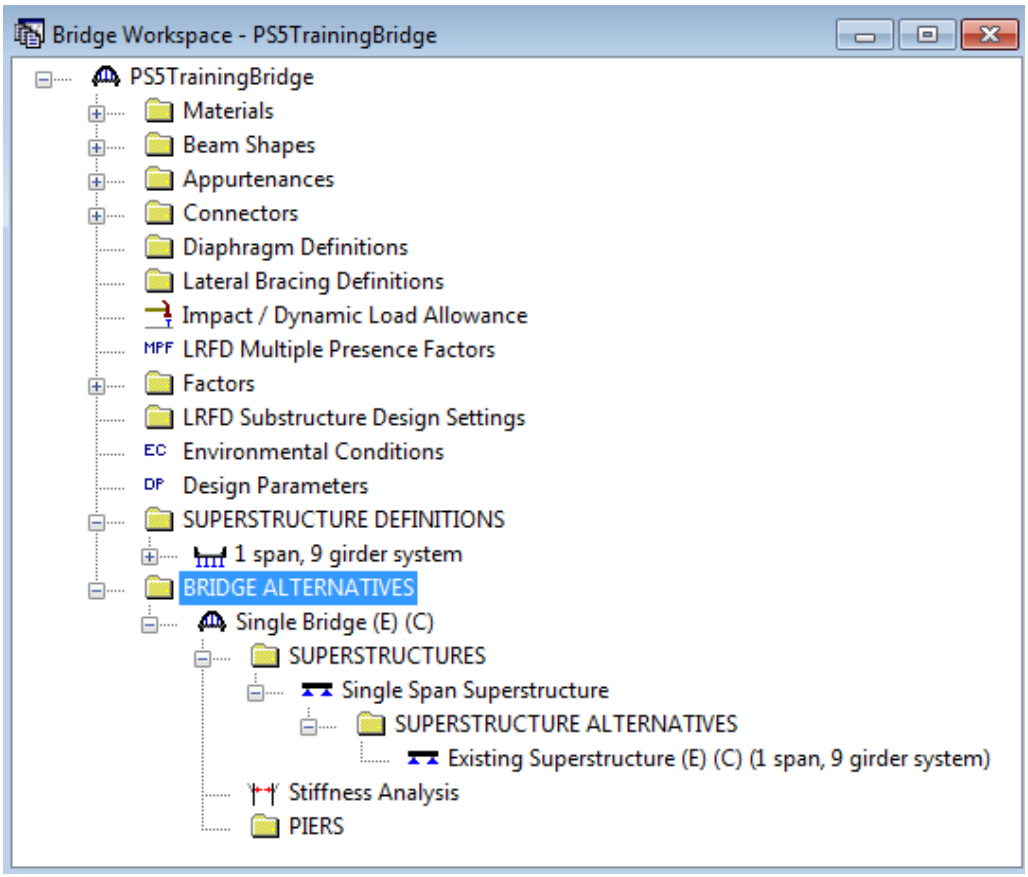


Select Girder System and the Structure Definition window will open. Enter the appropriate data as shown below:

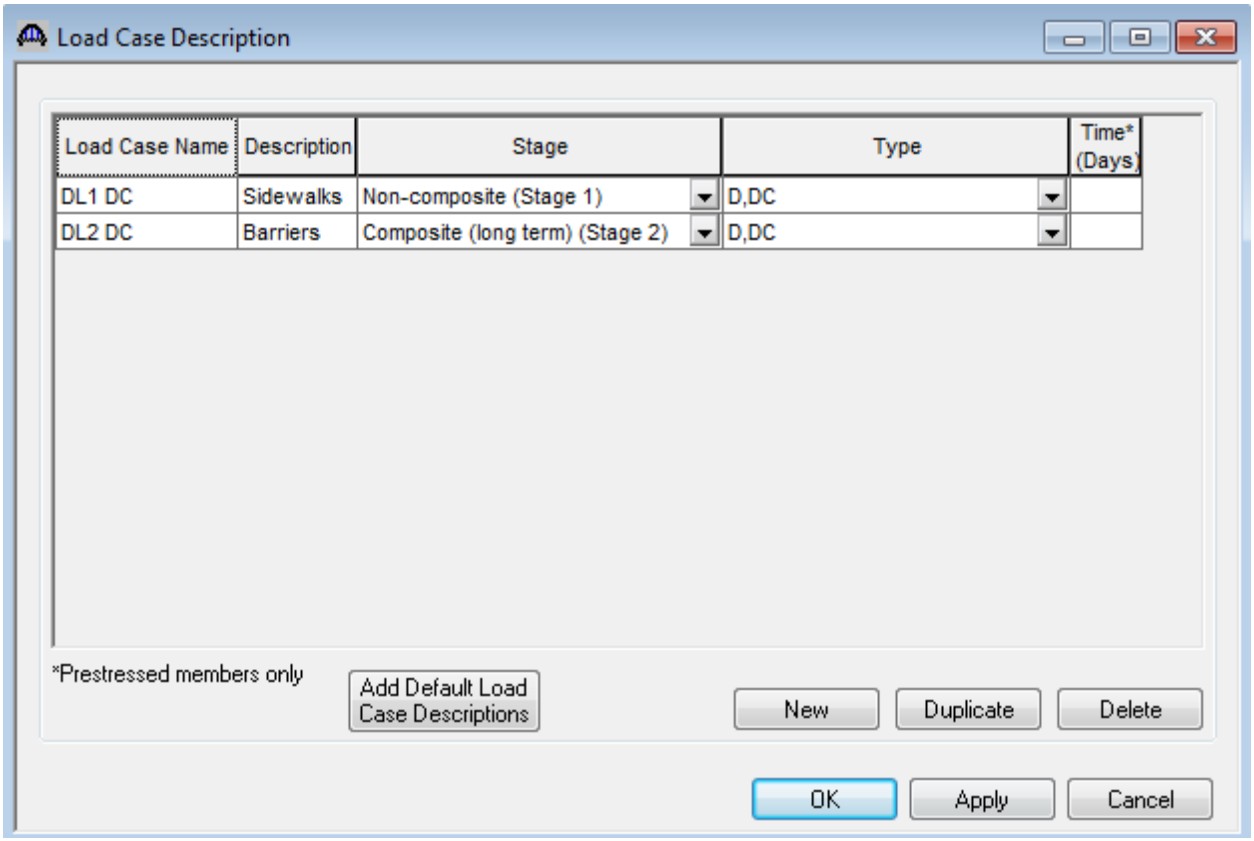


Click on Ok to save the data to memory and close the window.

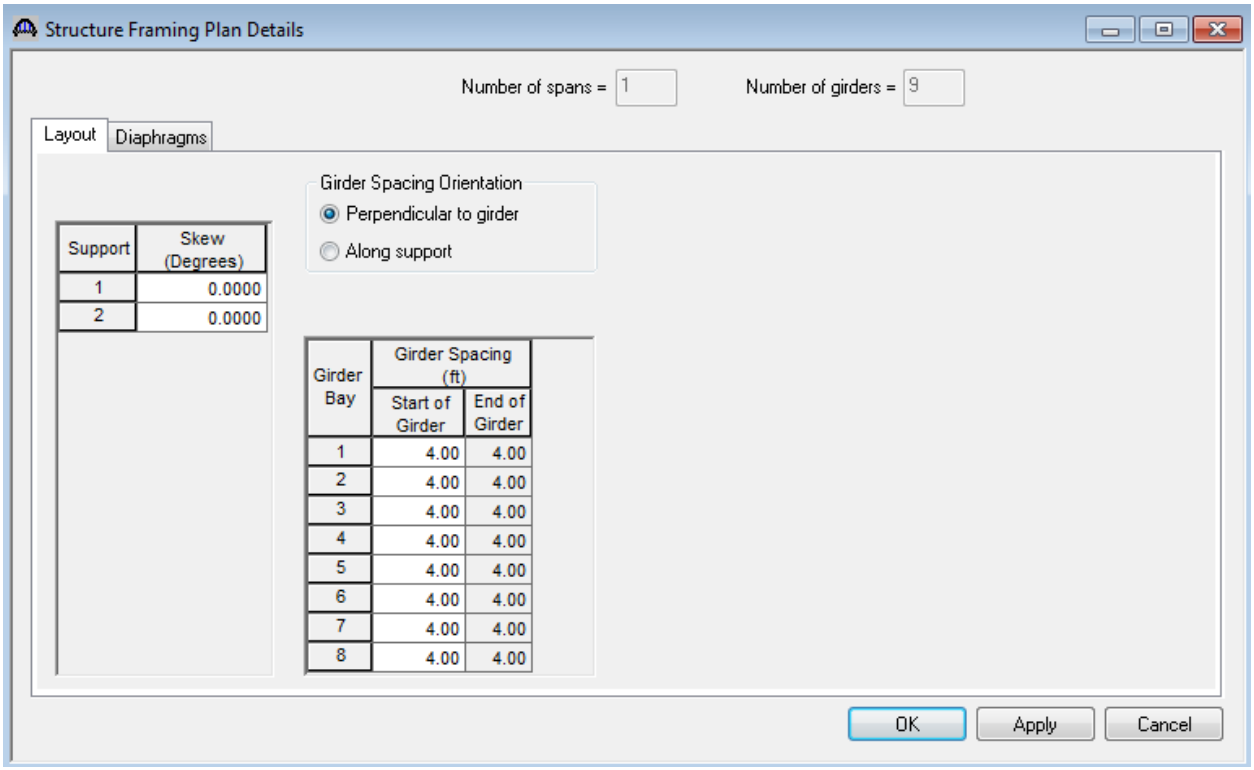
We now go back to the Bridge Alternatives and create a new Bridge Alternative, a new Superstructure, and a new Superstructure Alternative. The partially expanded Bridge Workspace tree is shown below:



Click Load Case Description to define the dead load cases. The completed Load Case Description window is shown below.

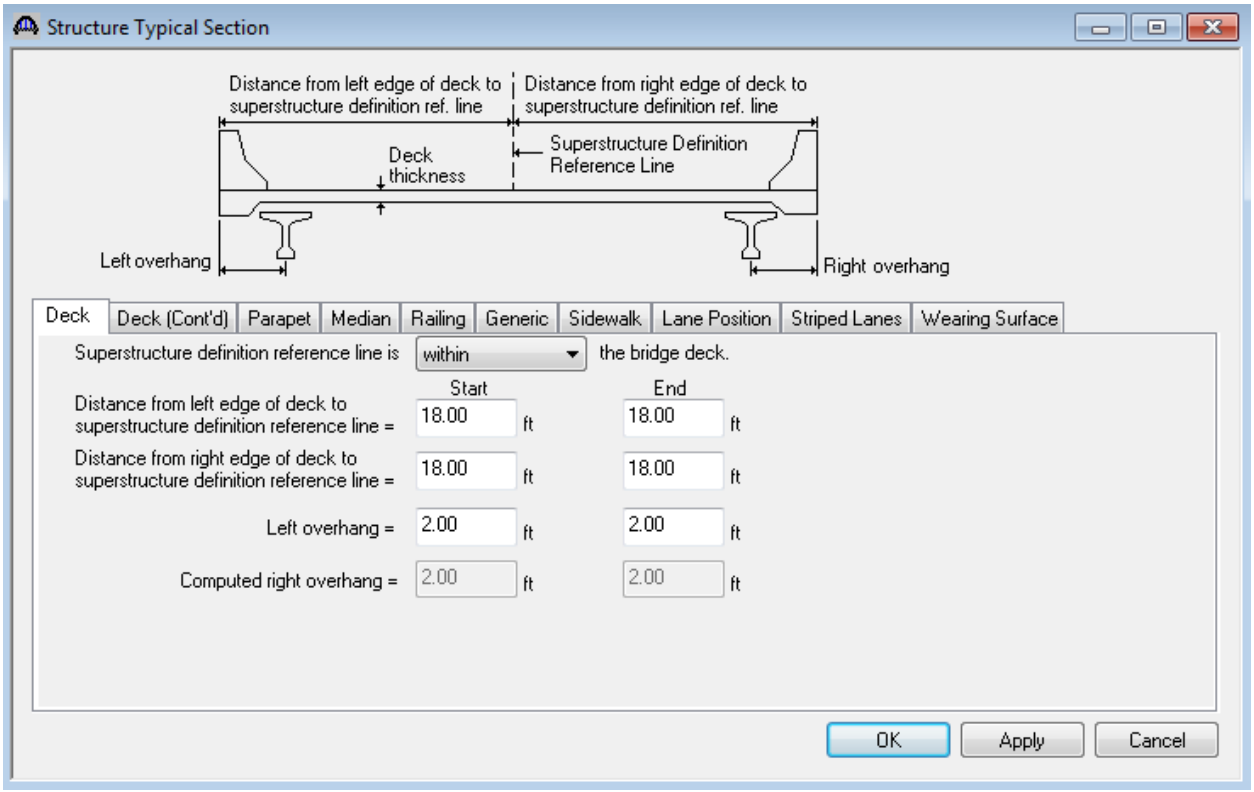


Double-click on Framing Plan Detail to describe the framing plan. Enter the appropriate data as shown below.

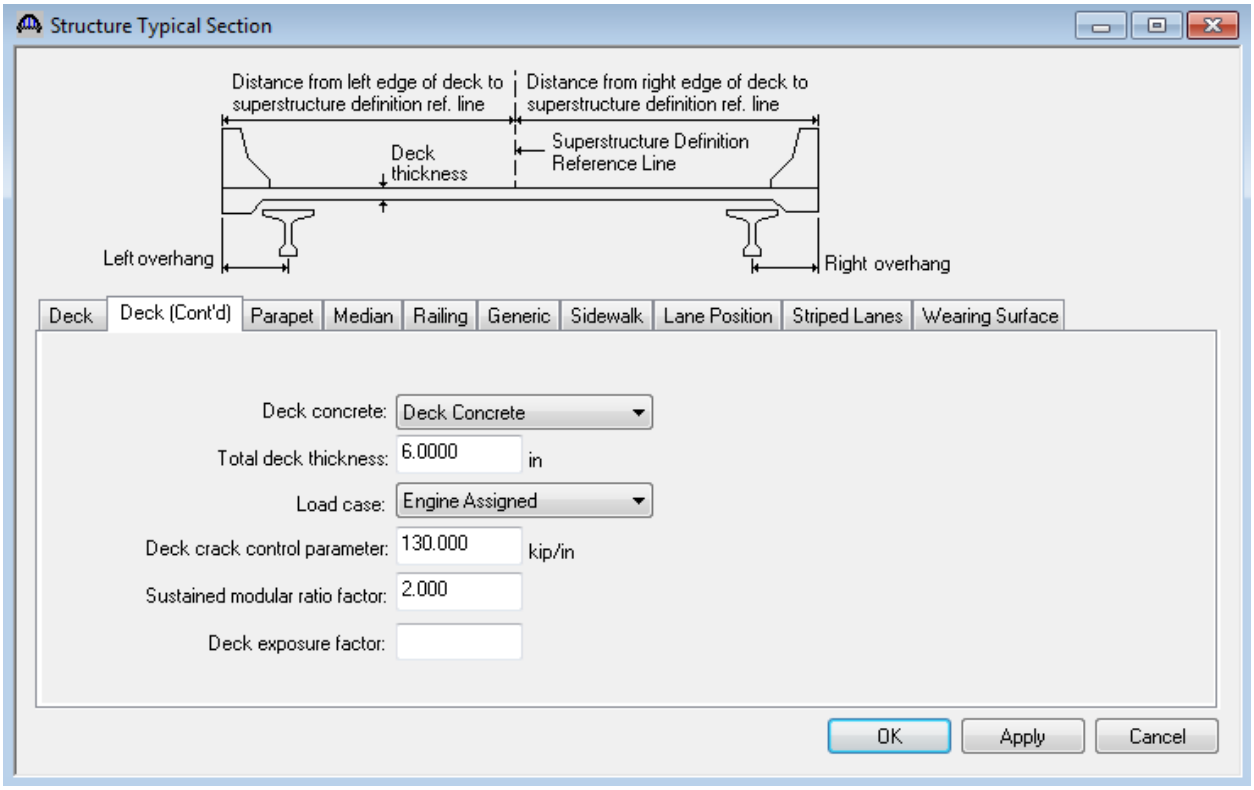


This structure does not have any external diaphragms since it is an adjacent box beam system. Click Ok to save the data to memory and close the window.

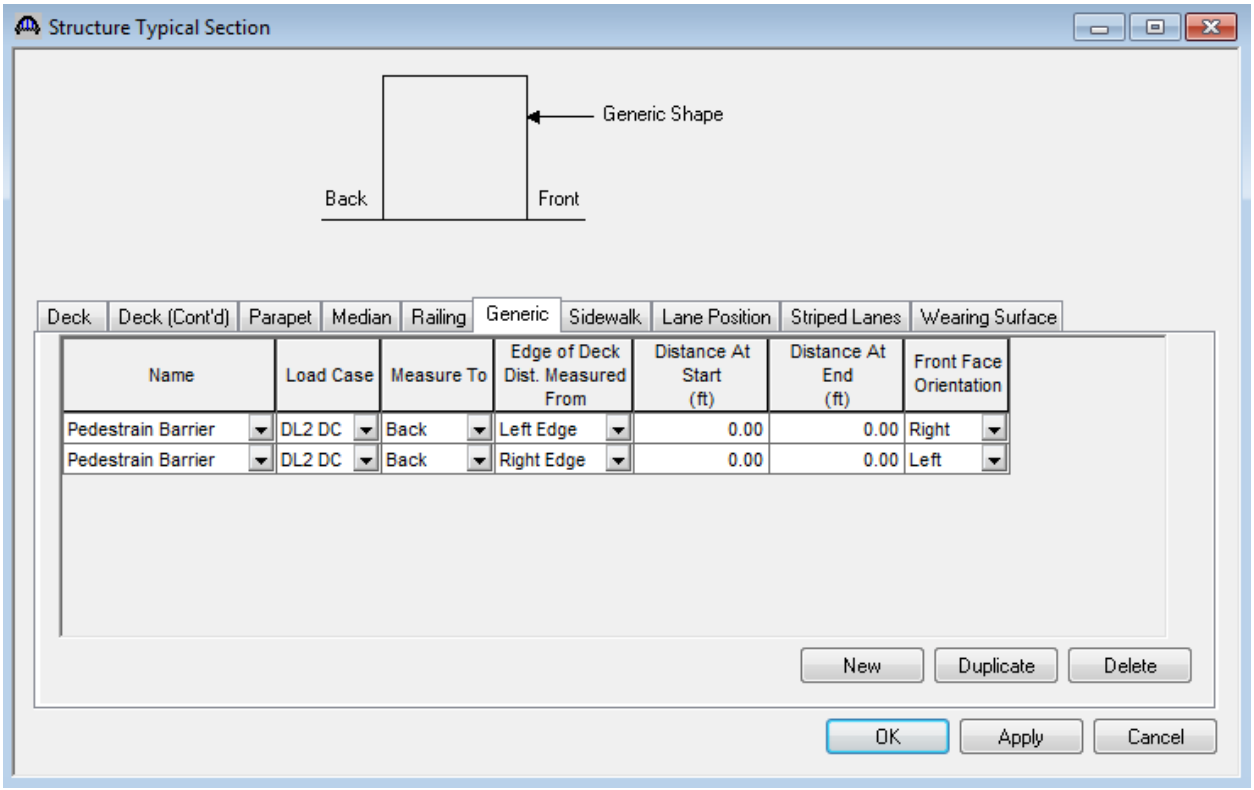
Next define the structure typical section by double-clicking on Structure Typical Section in the Bridge Workspace tree. Input the data describing the typical section as shown below.

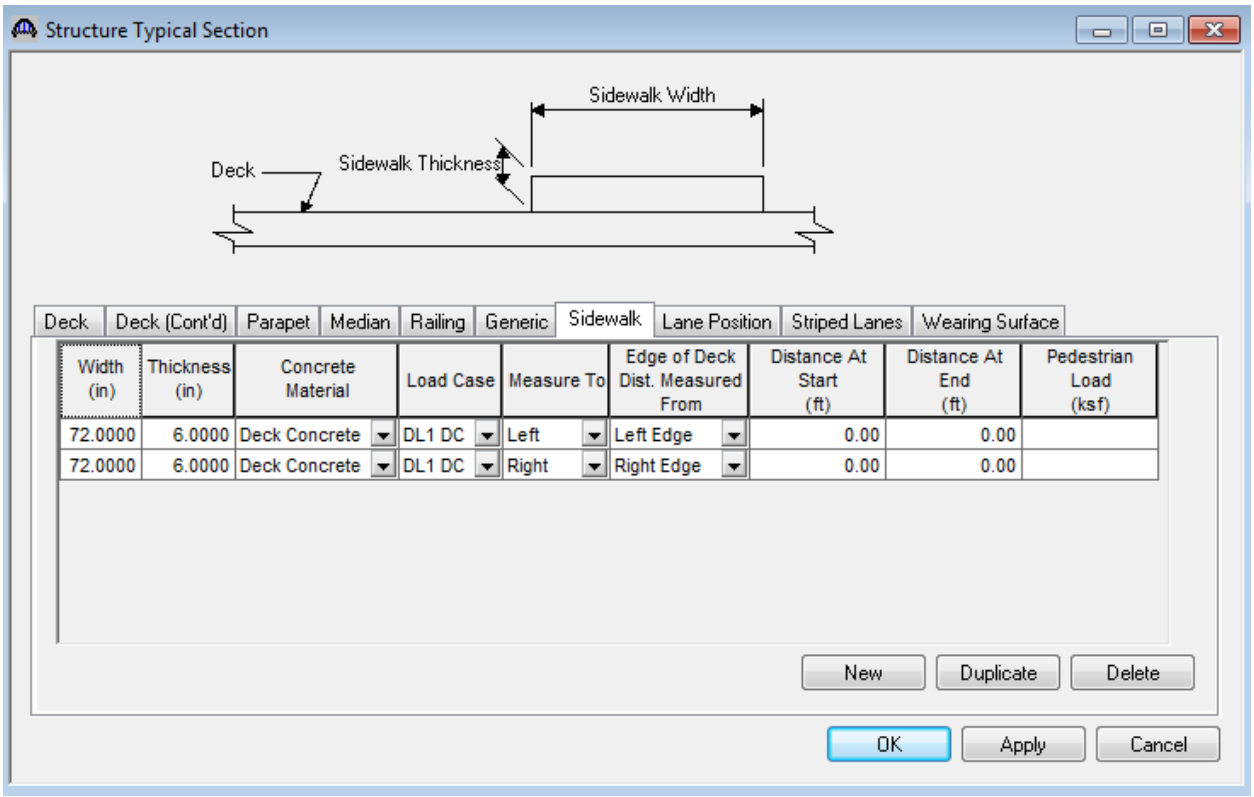


The Deck (cont'd) tab is used to enter information about the deck concrete and thickness. The material to be used for the deck concrete is selected from the list of bridge materials described above.



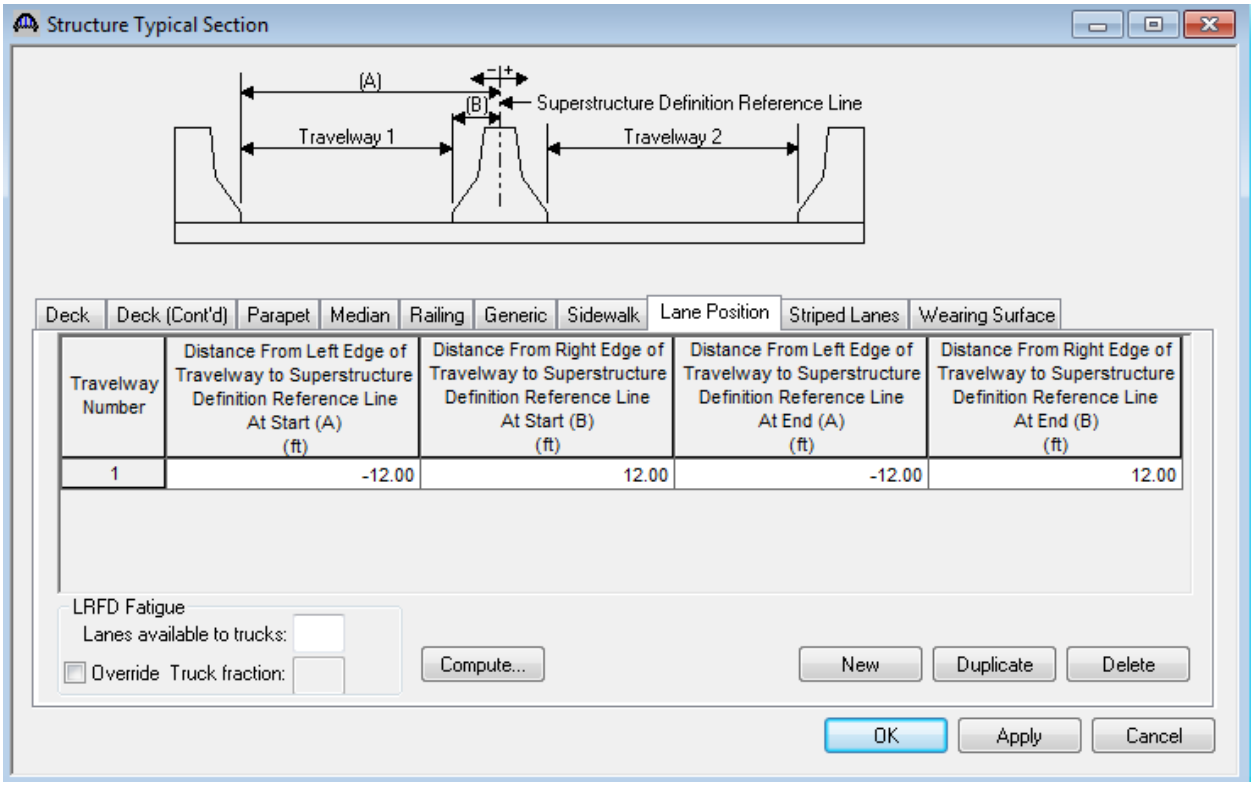
Add the concrete railings and sidewalk as follows:





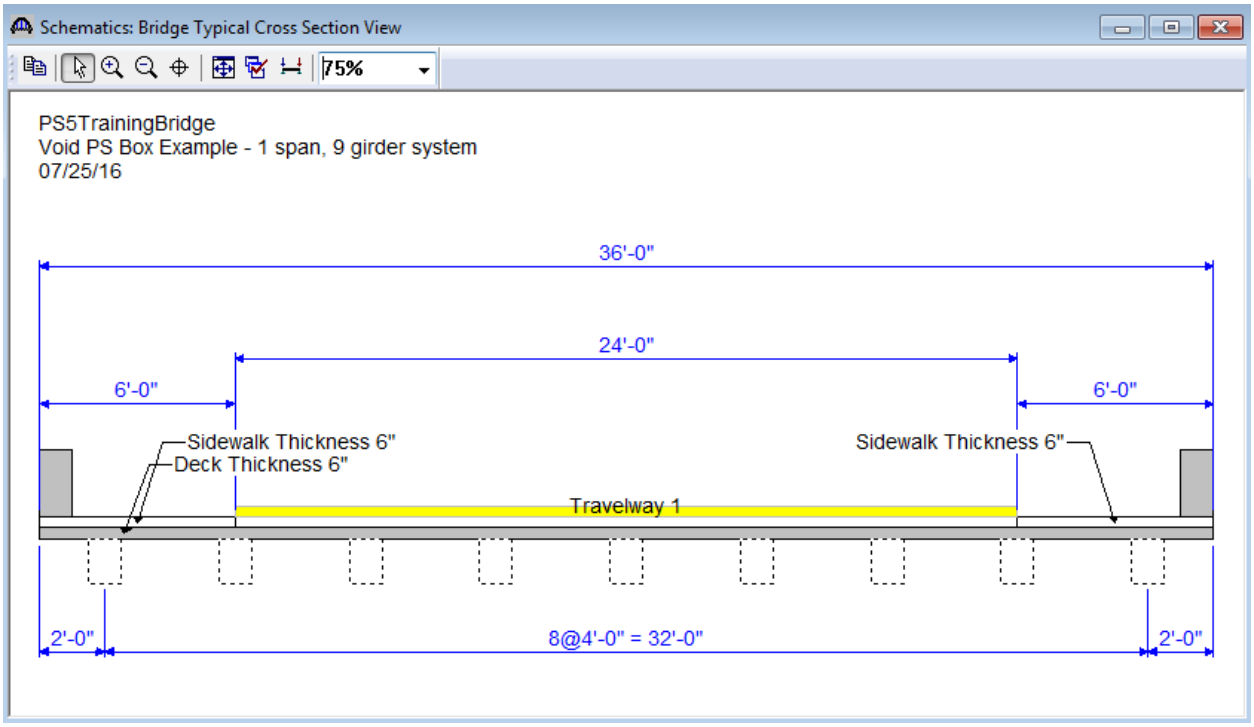
Lane Positions:

Select the Lane Position tab and use the Compute… button to compute the lane positions. A dialog showing the results of the computation opens. Click Apply to apply the computed values. The Lane Position tab is populated as shown below.



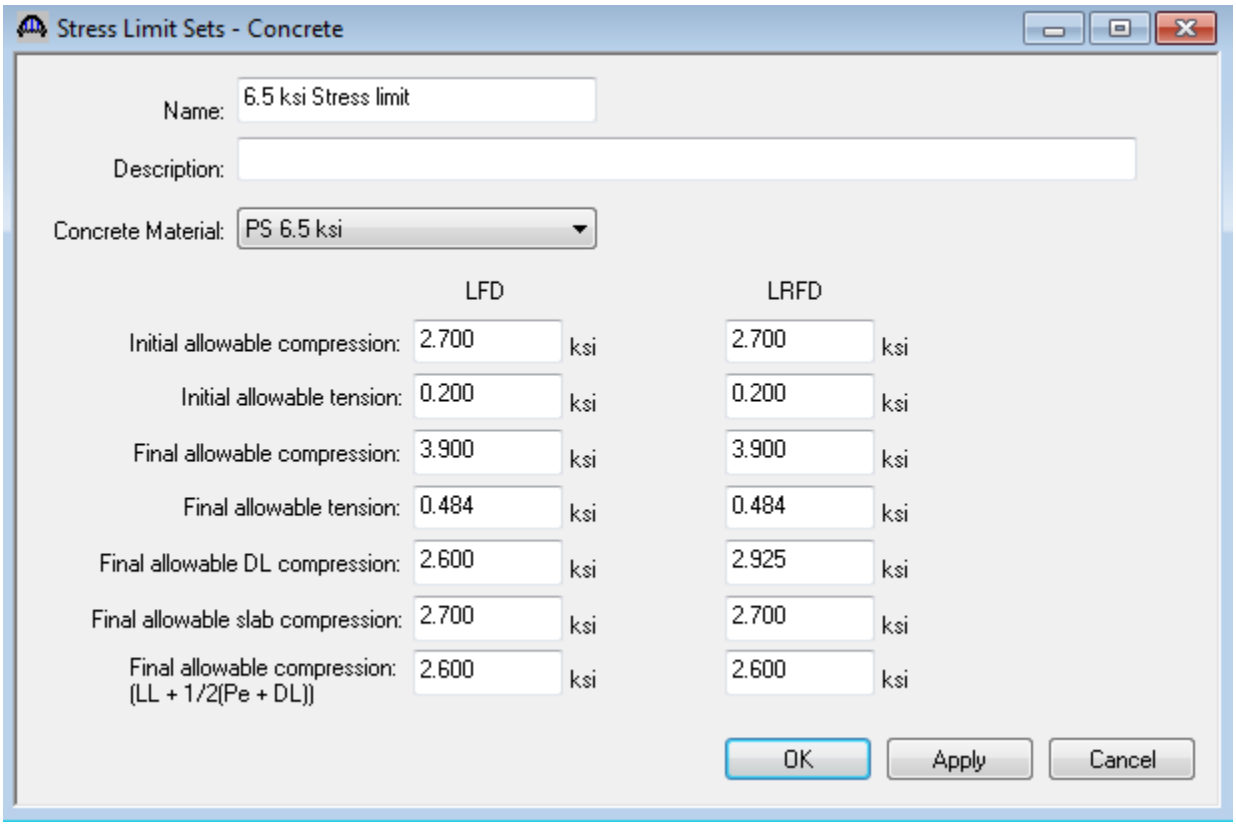
Click Ok to save the data to memory and close the window.

The structure typical section schematic can be viewed by selecting “Structure Typical Section” in the Bridge Workspace tree and clicking the “View Schematic” toolbar button.



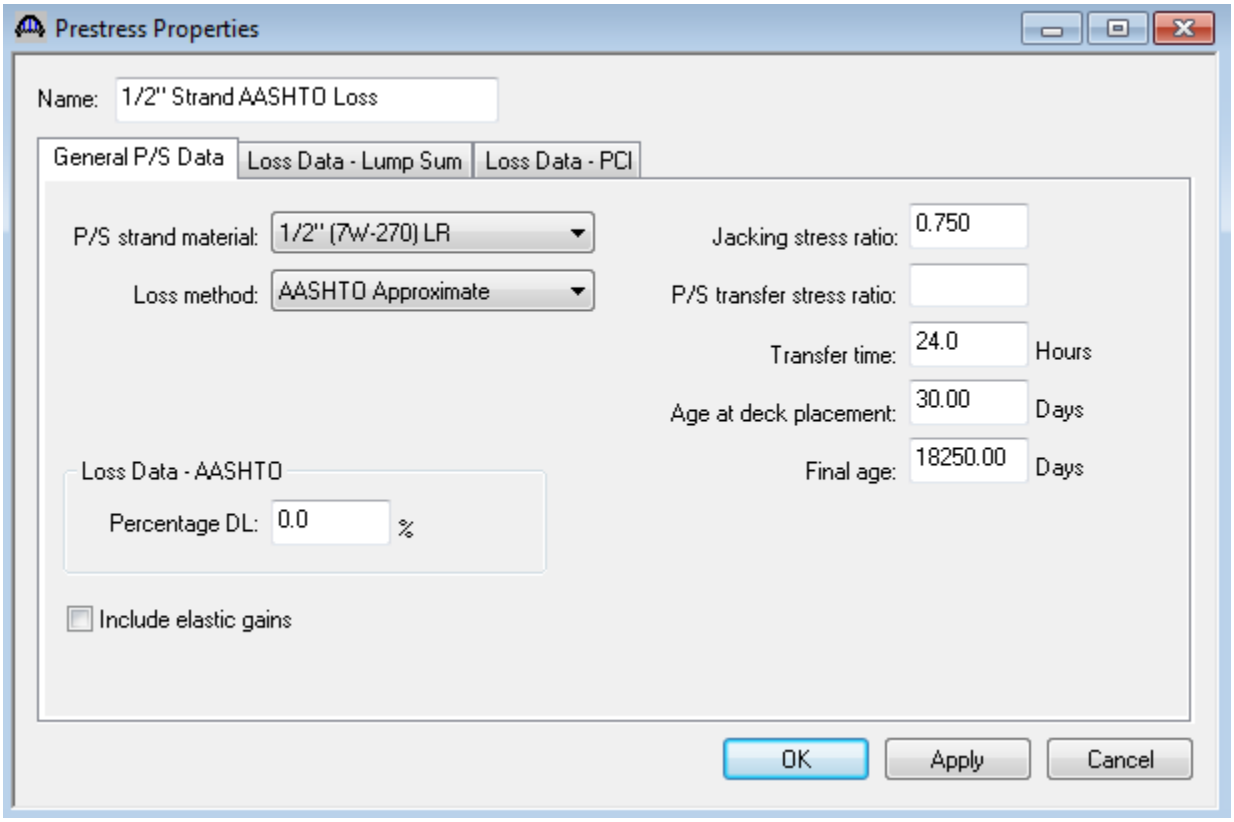
The beams are displayed as dashed boxes since we have not defined the beams yet.

Now define a Stress Limit. A Stress Limit defines the allowable concrete stresses for a given concrete material. Double click on the Stress Limits tree item to open the window. Select the "PS 6.5 ksi" concrete material. Default values for the allowable stresses will be computed based on this concrete and the AASHTO Specifications. A default value for the final allowable slab compression is not computed since the deck concrete is typically different from the concrete used in the beam.

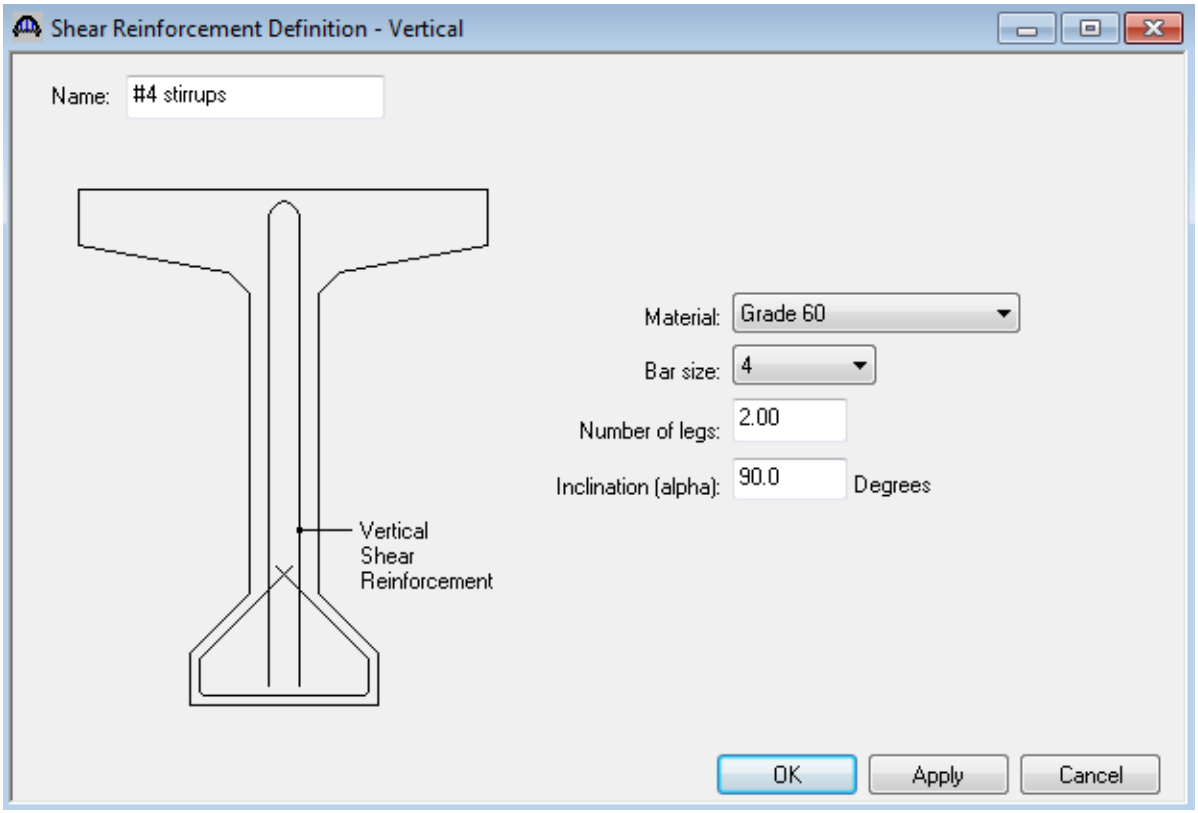


The final allowable tension values are calculated using the Stress Limit Coefficient found on the System Defaults: Bridge Workspace window. This coefficient is dependent on the moderate or severe corrosive condition to which the members are exposed.

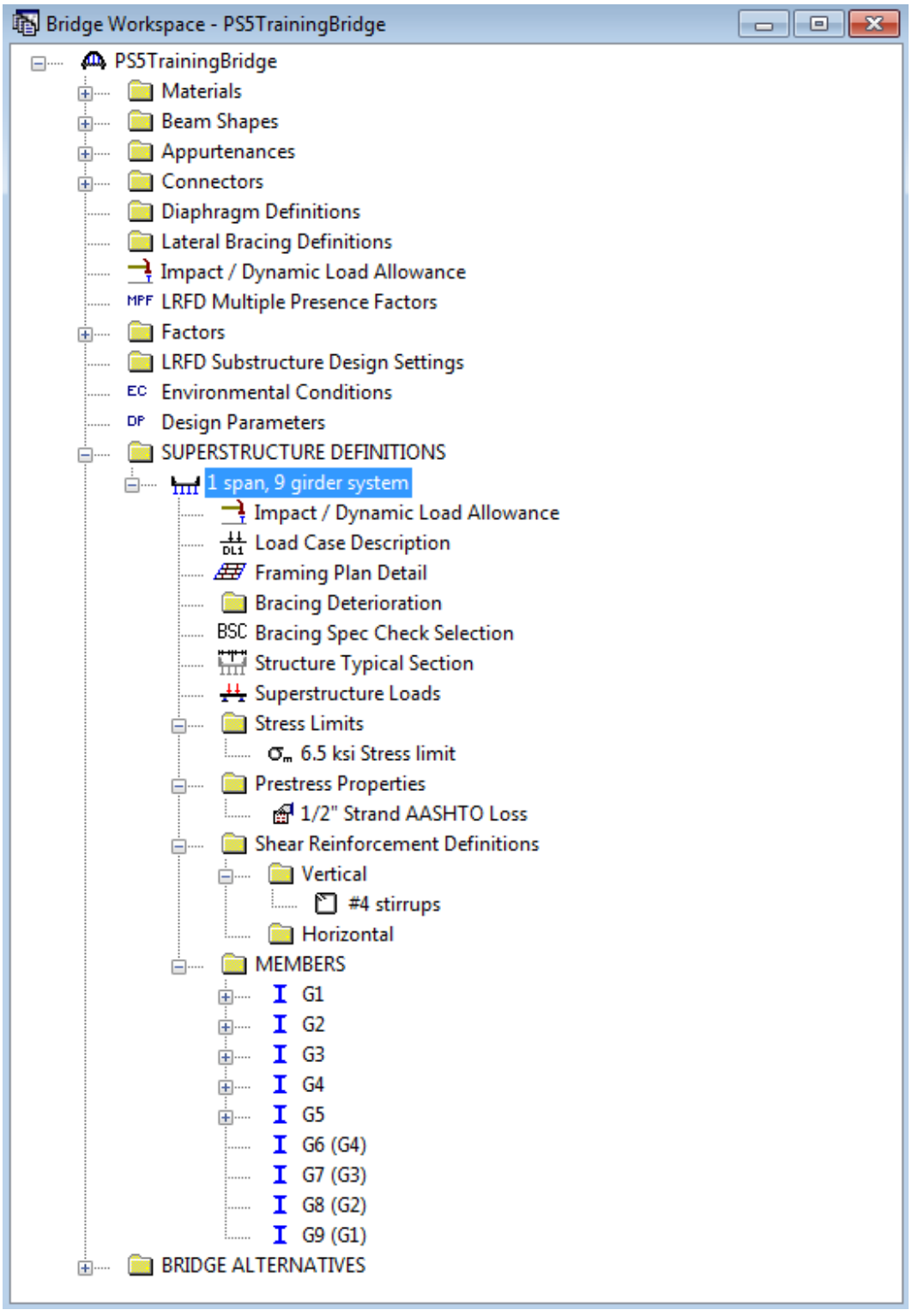
Double click on the Prestress Properties tree item to open a window in which to define the prestress properties for this structure definition. Define the Prestress Property as shown below. We are using the AASHTO method to compute losses so the "General P/S Data" tab is the only tab that we have to visit. Click Ok to save to memory and close the window.



Now define the vertical shear reinforcement by double clicking on Vertical (under Shear Reinforcement Definitions in the tree). Define the reinforcement as shown below. Click Ok to save to memory and close the window.

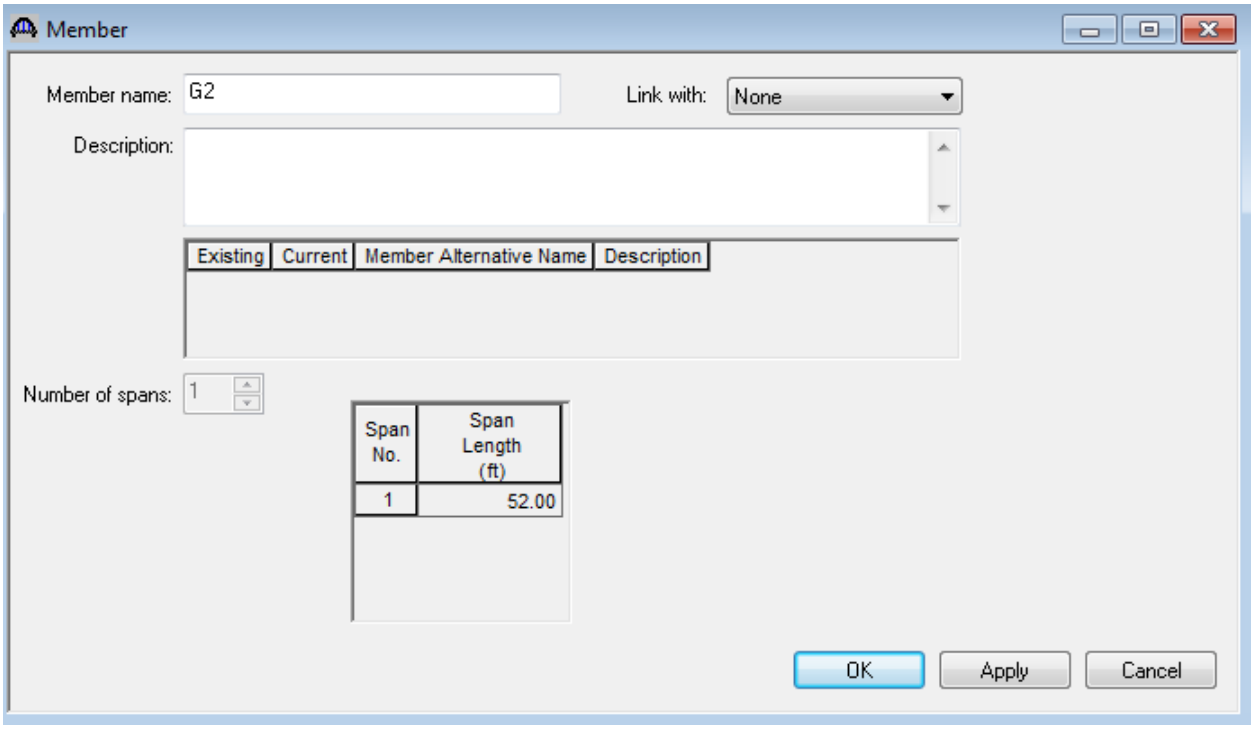


A partially expanded Bridge Workspace is shown below.



Describing a member:

The member window shows the data that was generated when the structure definition was created. No changes are required at this time. The first Member Alternative that we create will automatically be assigned as the Existing and Current Member alternative for this Member.

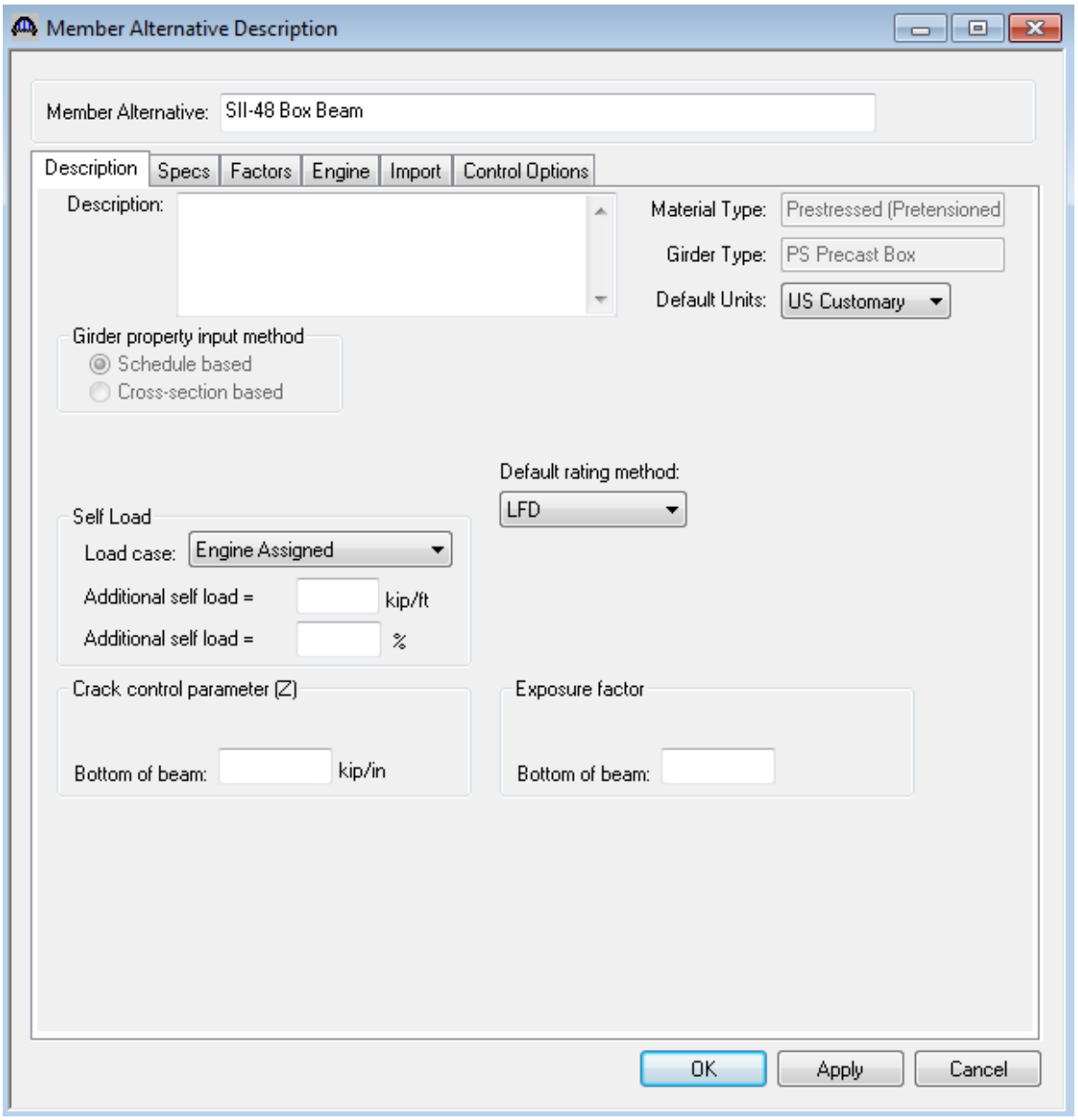


Defining a Member Alternative:

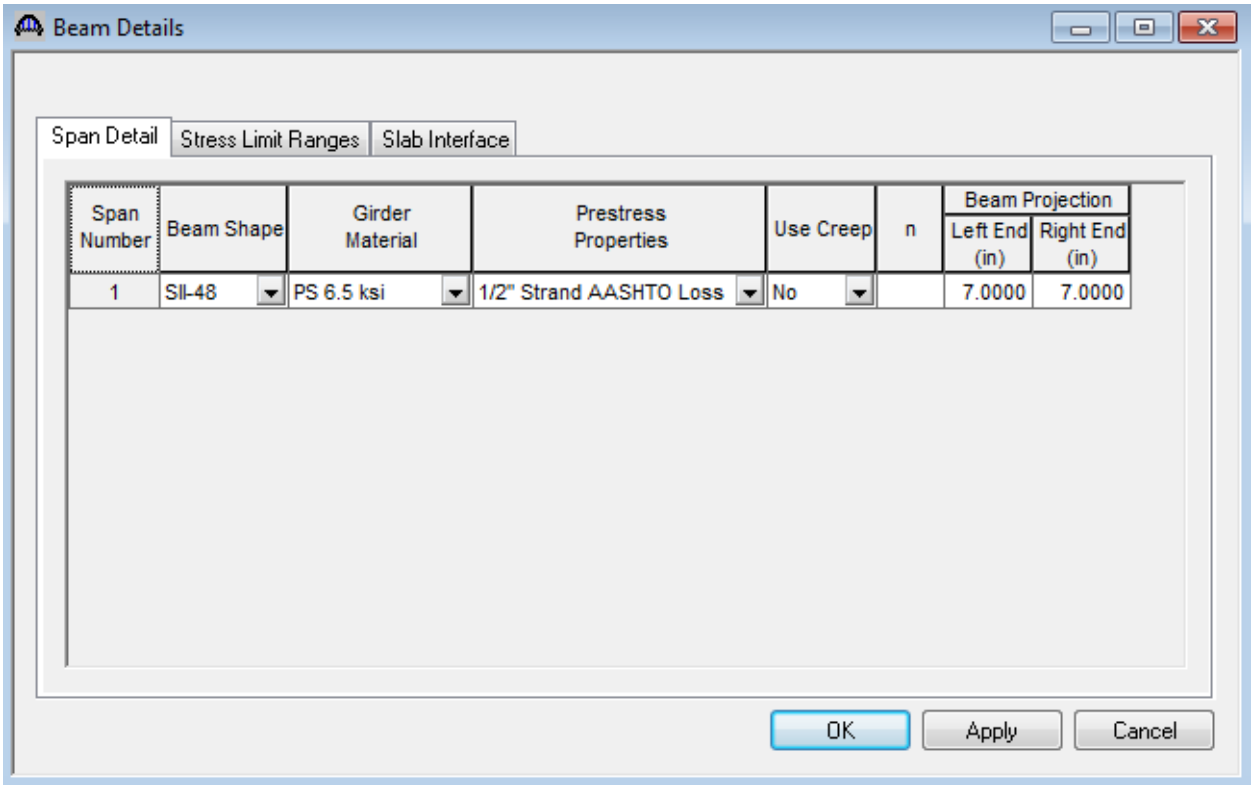
Double-click MEMBER ALTERNATIVES in the tree to create a new alternative. The New Member Alternative dialog will open. Select Prestressed (Pretensioned) Concrete for the Material Type and PS Precast Box for the Girder Type.

Click Ok to close the dialog and create a new member alternative.

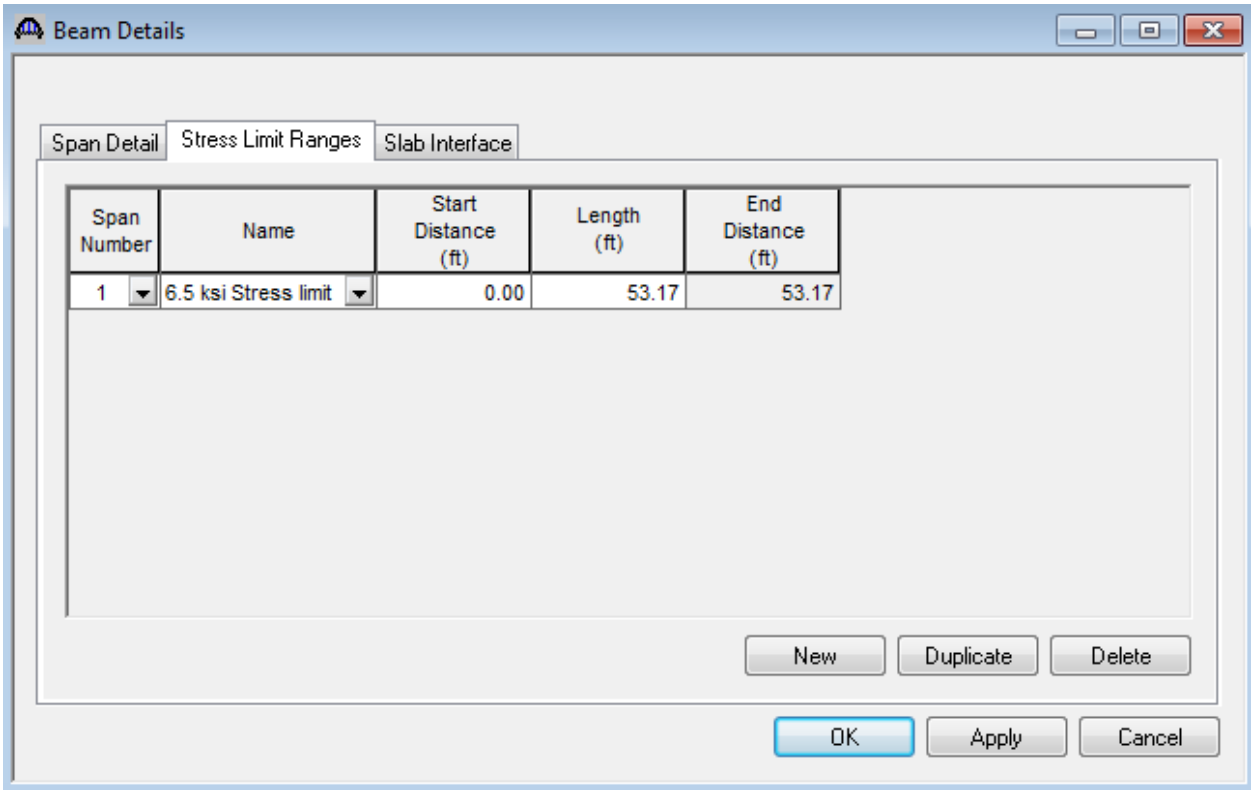
The Member Alternative Description window will open. Enter the appropriate data as shown below. The Schedule-based Girder property input method is the only input method available for a prestressed concrete beam.



Next describe the beam by double clicking on Beam Details in the tree. The beam shape must be selected in this window prior to using the Compute from Typical Section button in the Live Load Distribution Factors window. So far we have identified the member alternative as being a prestressed box but we have not yet defined if it is a spread box or an adjacent box. The calculation of the live load distribution factors is dependent on the boxes being spread or adjacent. Once we select the beam shape in the Beam Details window, BrR will be able to determine if the boxes are spread or adjacent based on the box width and the girder spacing entered earlier. The Beam Details windows with the appropriate data are shown below.

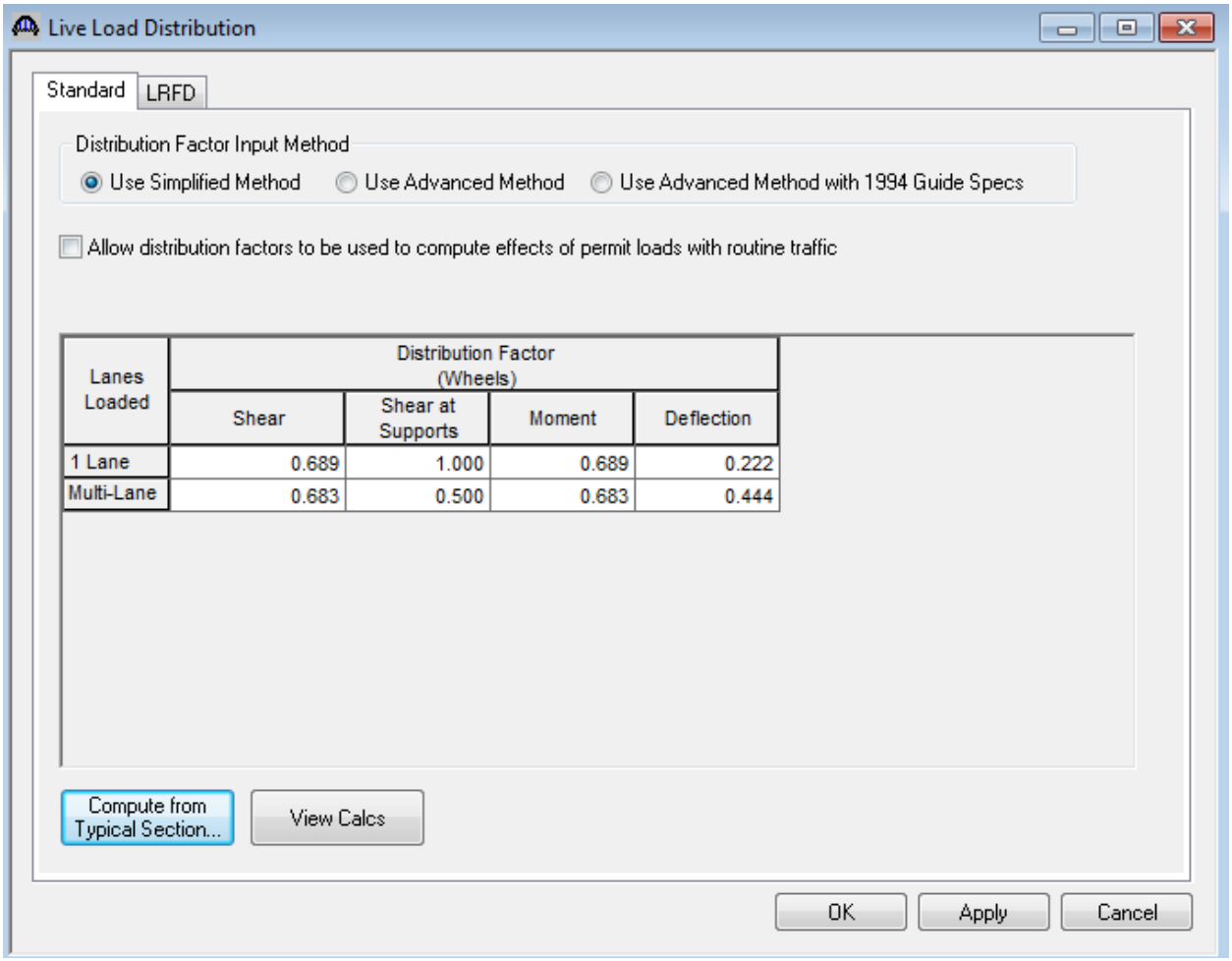


Note that Stress Limit Ranges are defined over the entire length of the precast beam, including the projections of the beam past the centerline of bearing which were entered on the Span Detail tab.

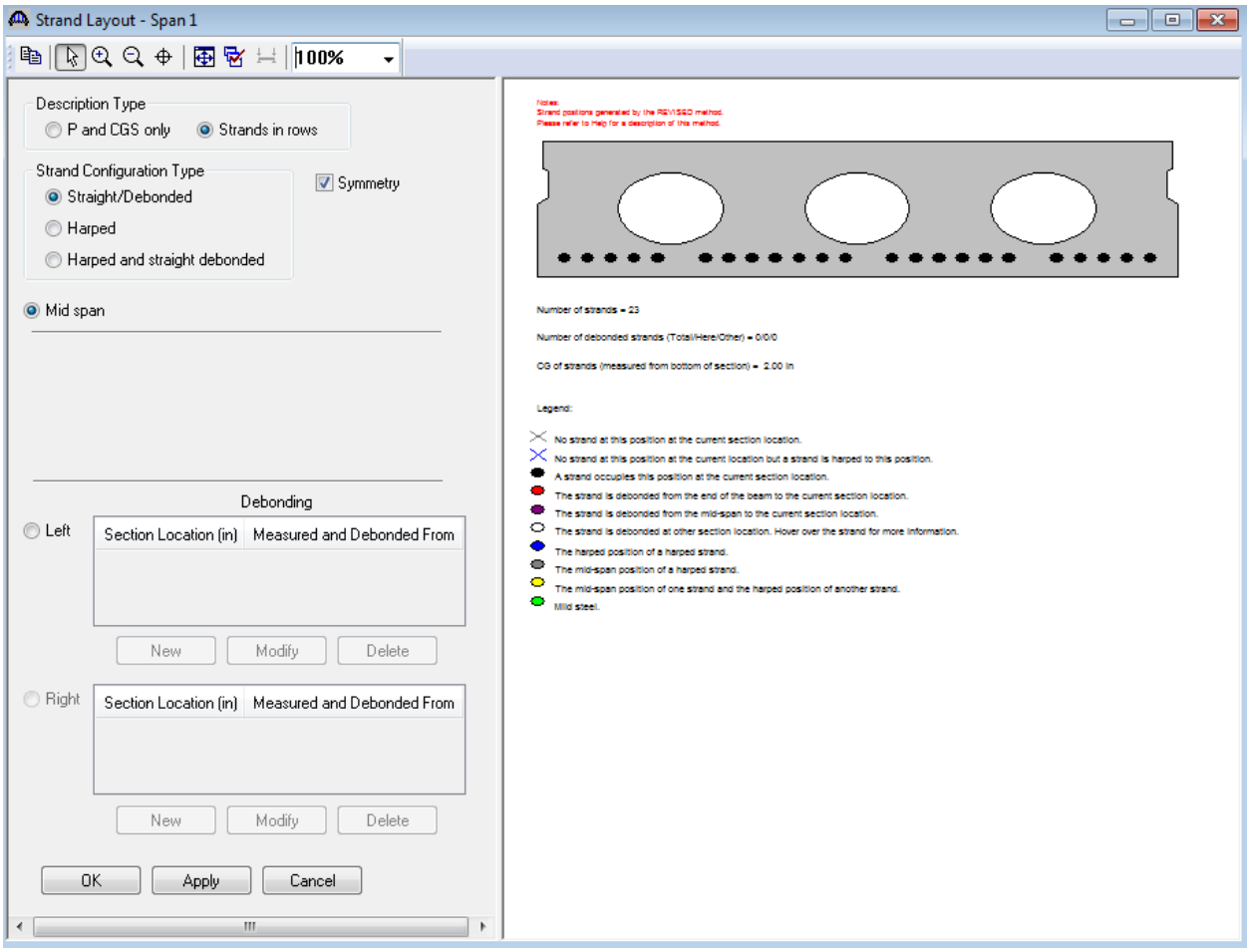


The defaults on the Slab Interface tab are acceptable. Click Ok to save the Beam Details data to memory and close the window.

Open the Live Load Distribution window. On the Standard tab click on the Compute from Typical Section button to populate the LFD live load distribution factors based on the Structure Typical Section. BrR/BrD will compute the distribution factors based on the girder type, girder spacing, deck geometry and lane positions as per the AASHTO Standard Specifications for Highway Bridges.

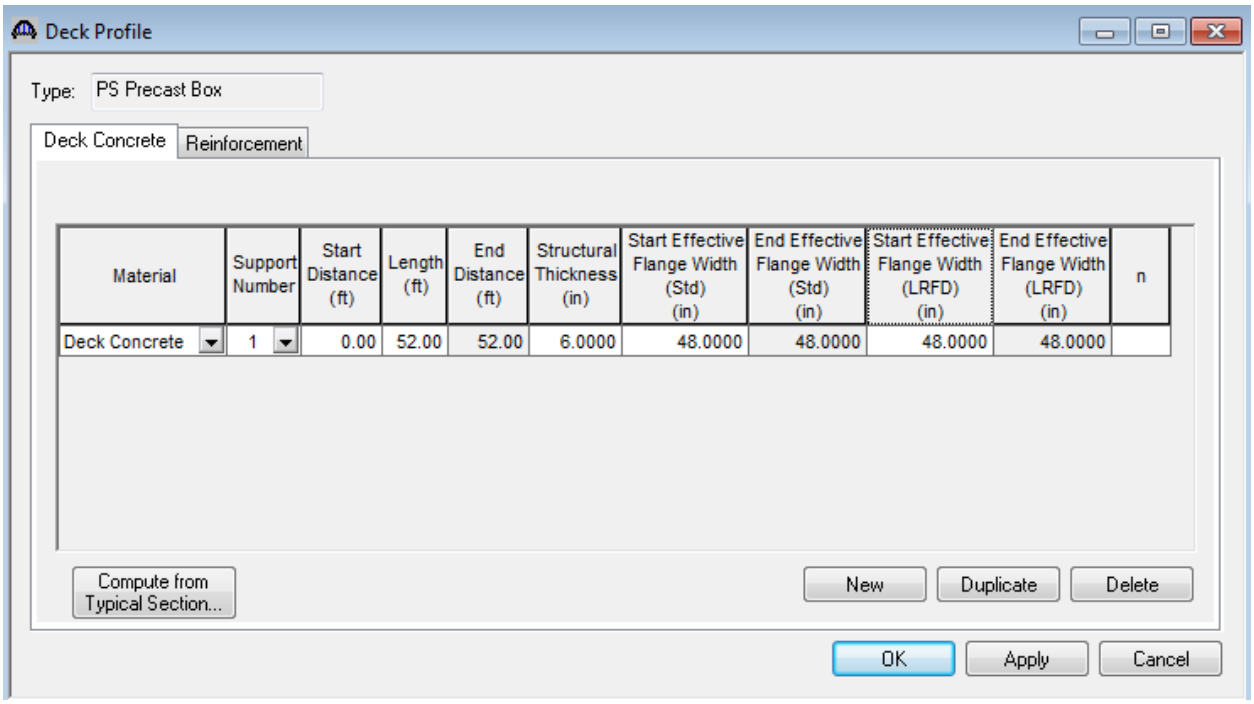


Expand the Bridge Workspace tree under Strand Layout and open the Span 1 window. Select the Description Type as “Strands in Rows” and the Strand Configuration Type as “Straight/Debonded”. Enter the following data to describe the prestress strand configuration. There is no debonding to describe for this beam.



Click Ok to save the data to memory and close the window.

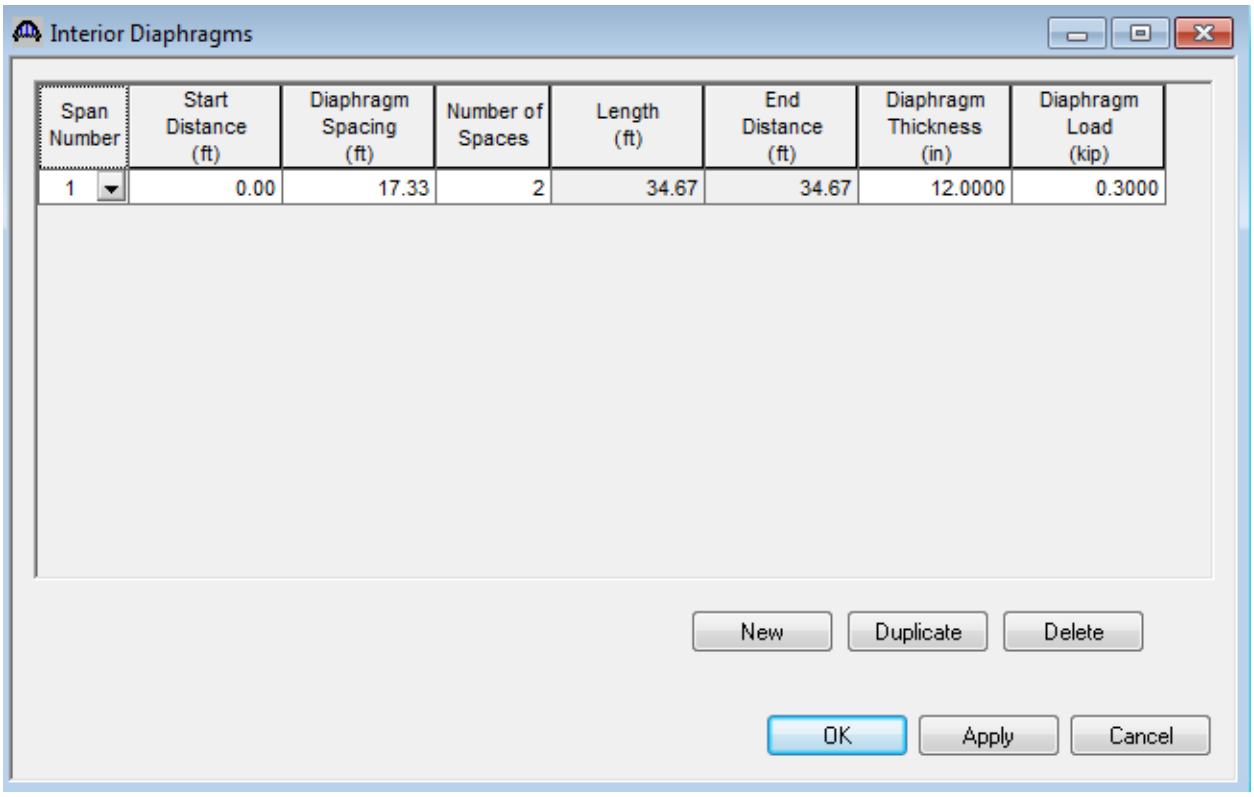
Next open the Deck Profile and enter the data describing the structural properties of the deck. The window is shown below.



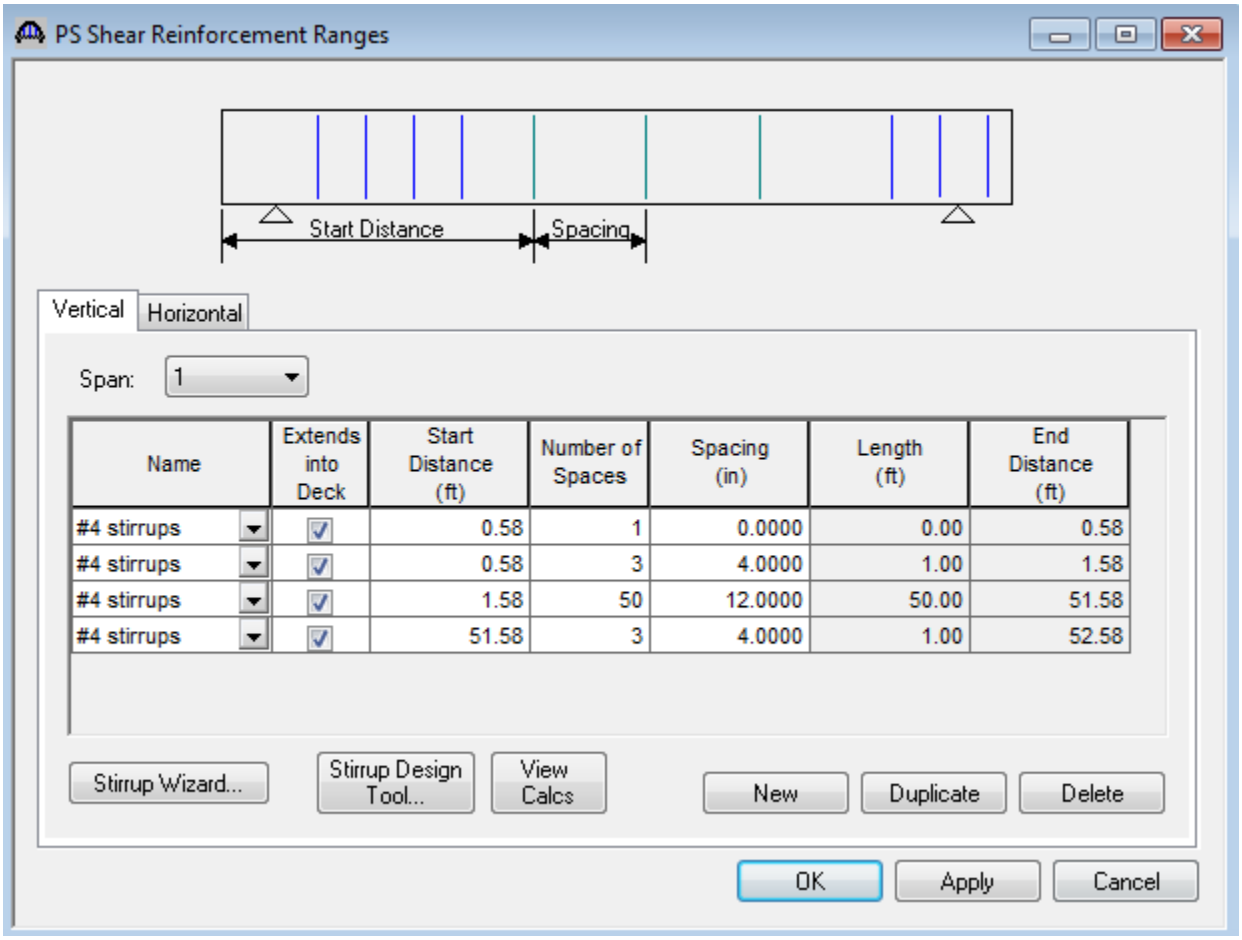
No reinforcement is described.

There is no haunch profile to define either.

The internal diaphragms for this box member are entered as described below.

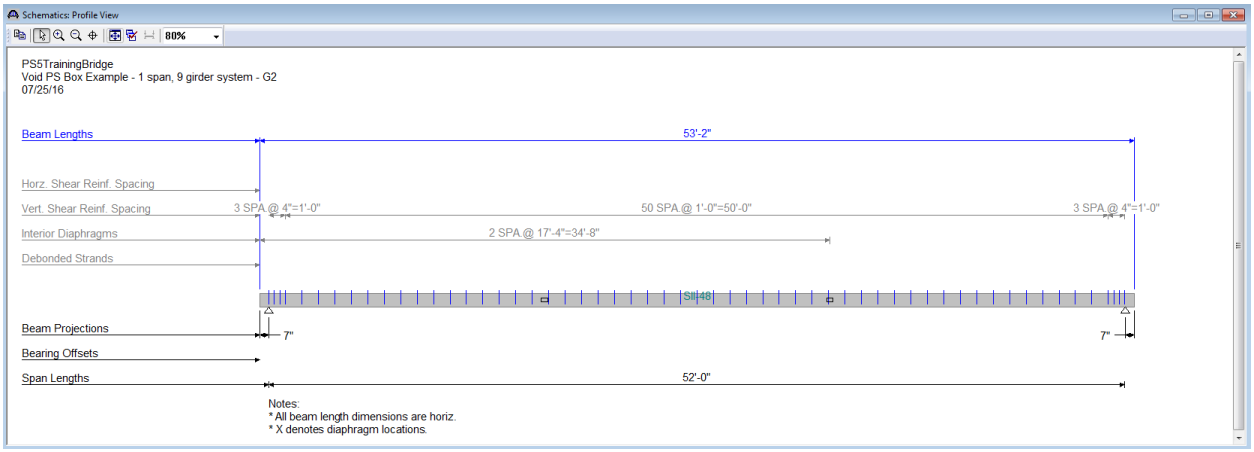


The Shear Reinforcement Ranges are entered as described below. The vertical shear reinforcement is defined as extending into the deck on this tab. This indicates composite action between the beam and the deck. Data does not have to be entered on the Horizontal tab to indicate composite action since we have defined that by extending the vertical bars into deck.

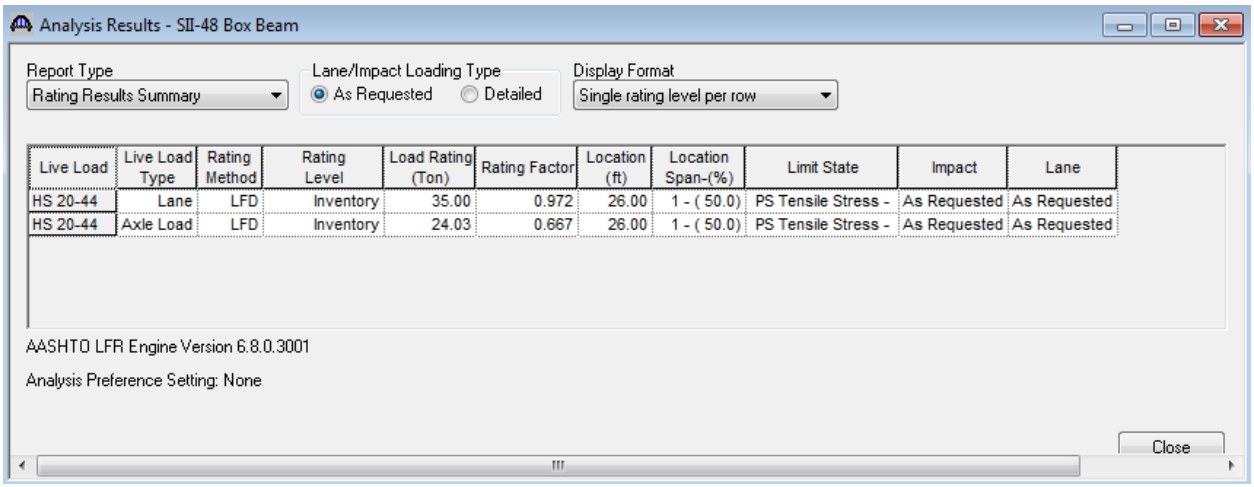


The description of an interior beam for this structure definition is complete.

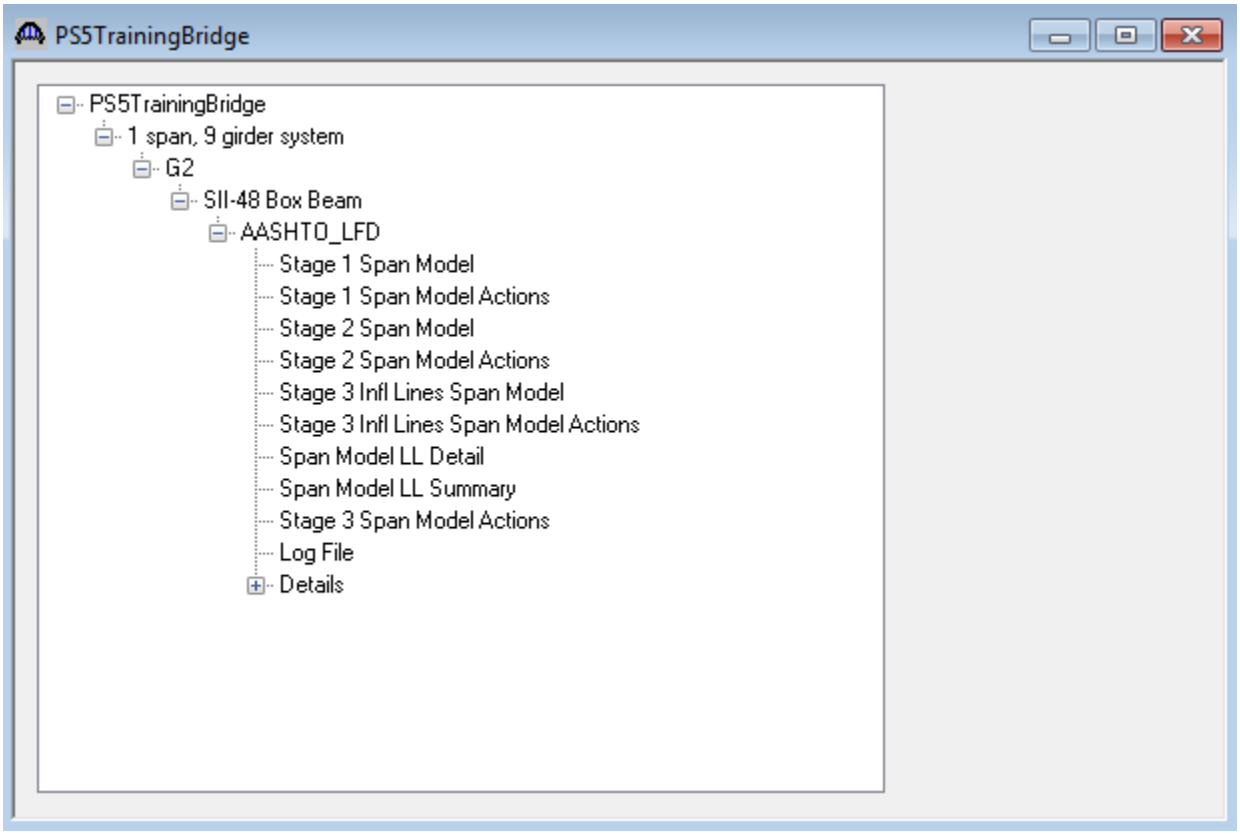
The girder elevation can be displayed by selecting the “View schematic” toolbar button while the name of the member alternative is selected in the Bridge Workspace tree.



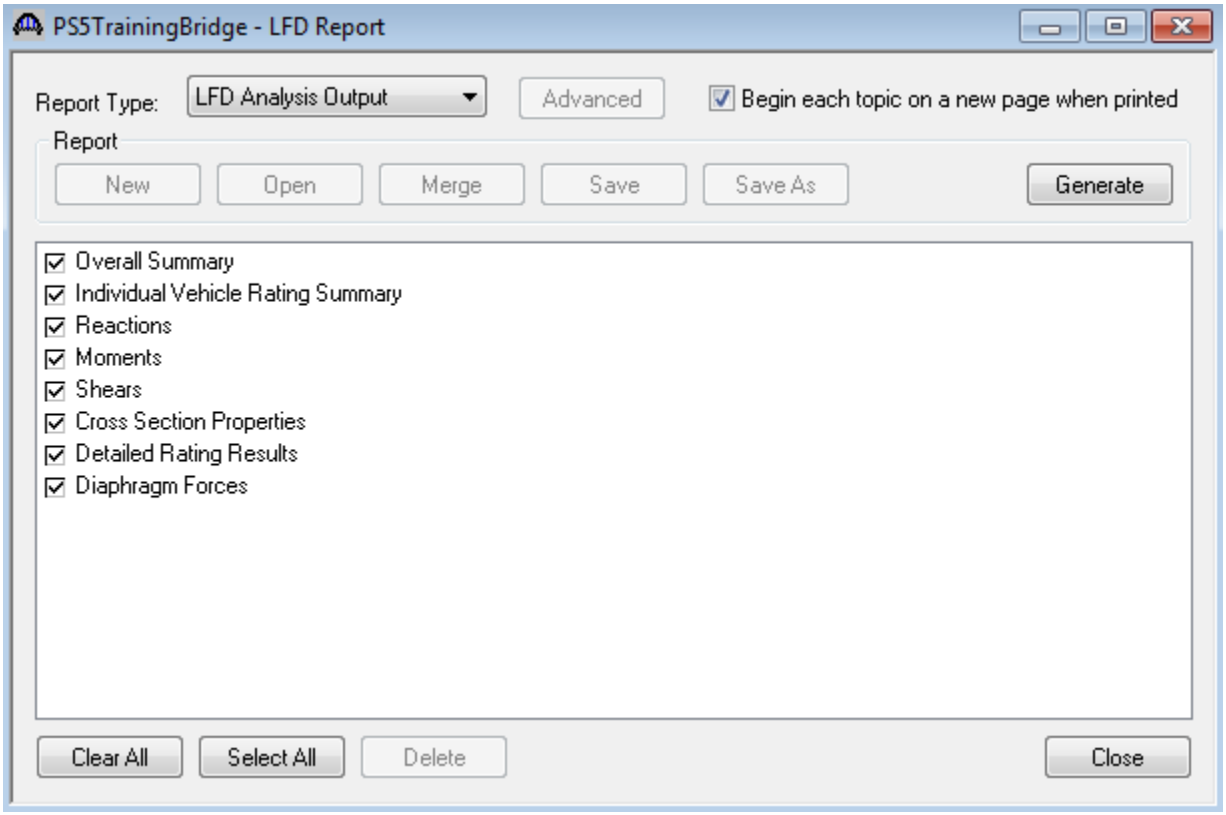
The results for rating an HS20 vehicle using Load Factor Design are shown below:



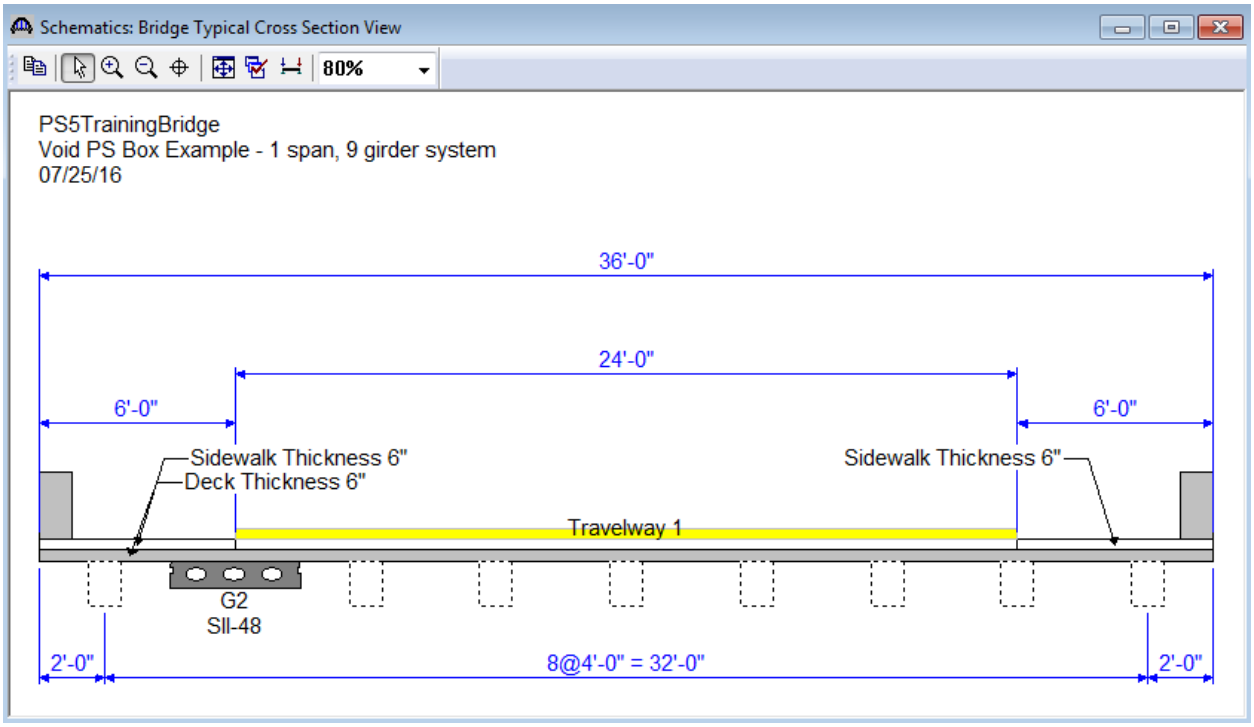
The analysis output files can be viewed by selecting the “View analysis output” button on the toolbar. The following window will open.



A summarized report of the output can be generated by selecting the “Report Tool” button from the toolbar. Select “LFD Analysis Output” in the Report Type box. You can specify which output you want the report to contain. Selecting “Generate” will open Internet Explorer and display the generated report. The report can be printed from Internet Explorer.



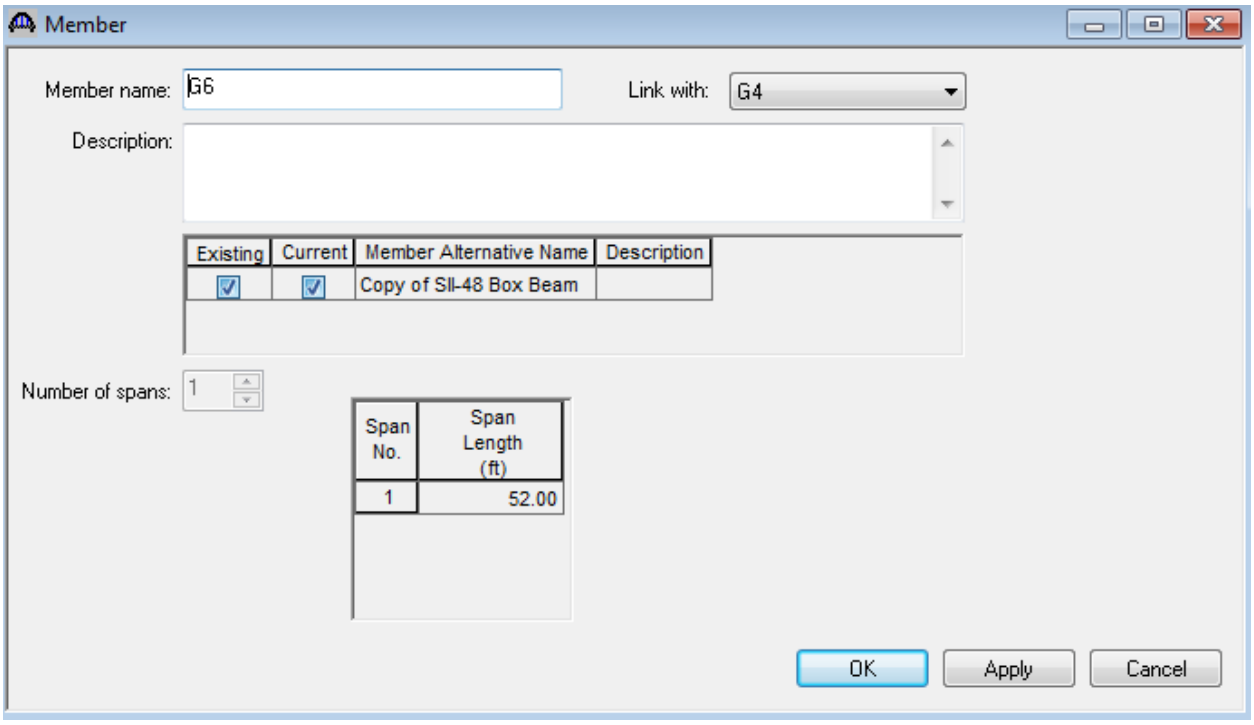
The following schematic is displayed if we select “View schematic” while the Structure Typical Section is selected in the Bridge Workspace tree.



Member G2 is the only member that has a member alternative defined. We can easily copy this member alternative to Member G1 to minimize the amount of data entry for this structure. Select “SII-48 Box Beam” under Member G2 in the Bridge Workspace tree and click the “Copy” button on the toolbar. Select “MEMBER ALTERNATIVES” under Member G1 and click the “Paste” button. Now Member G1 has a member alternative named “Copy of SII-48 Box Beam”. Since G1 is an exterior girder and G2 is an interior girder, there are a few windows under G1 that must be revisited to modify the data to represent an exterior girder. The Live Load Distribution Factors window should be opened and appropriate distribution factors entered for this exterior girder. The Deck Profile and Haunch Profile windows should also be modified to represent the exterior girder.

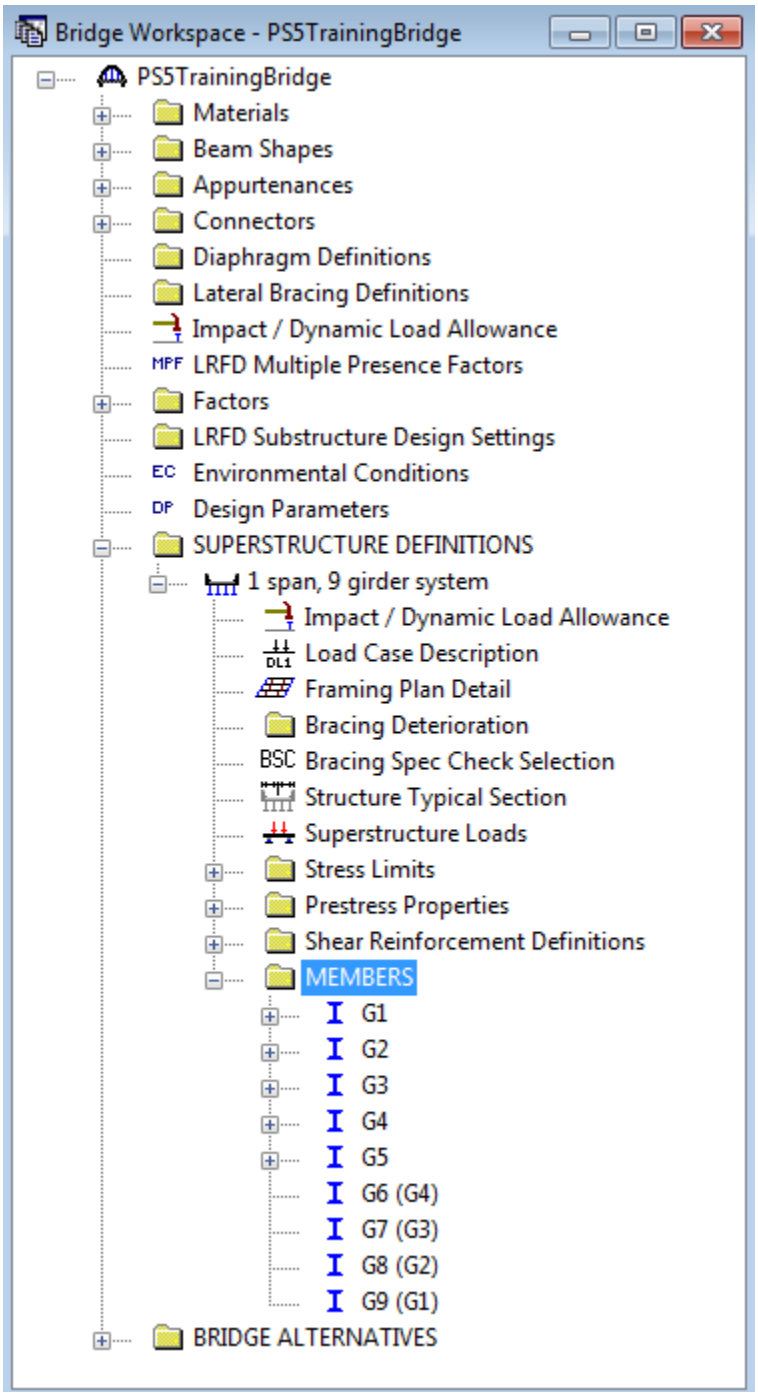
Use the copy function to copy the member alternative for Member G2 to Members G3, G4, and G5. Adjust the live load distribution factors as necessary for Members G3, G4, and G5. Depending on the arrangement of the lane positions and the girders, the simple beam distribution factors for the interior girders may differ. It is a good idea to recalculate the distribution factors for copied member alternatives. We now have half of the girders in the structure defined.

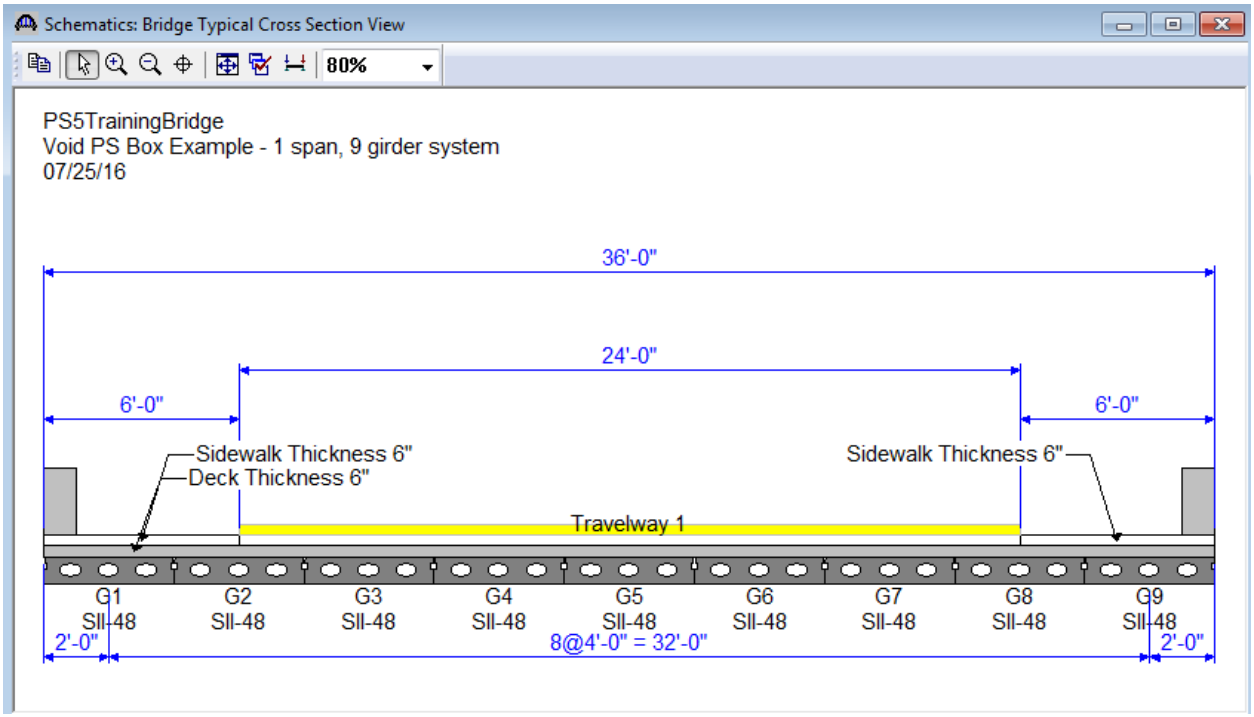
We can use the Link function in BrR to Link the remaining girders in the structure to the members we have defined. If two members are linked, they share the same definition and any revisions to one member affects the other member. If there are any differences between two members, then they should not be linked with one another. If the applied loads acting on the two members are different (due to different tributary widths, different arrangements of parapets, medians, sidewalks, and railings, and different lane positions), then they should not be linked with one another. All calculations are based on the properties and loads of the original girder. Open the Member G6 window and select "G4" in the Link with box. A warning message will appear to remind you that both members must share the exact same definition if they are to be linked. Click "Continue" to link the two members. Click Ok to save the data to memory and close the window.



Member G6 now shares the same definition as Member G4. Use this procedure to link Member G7 with G3, Member G8 with G2, and Member G9 with G1.

The following Bridge Workspace and Structure Typical Section Schematic will result.




Schematics: Bridge Typical Cross Section View
80%
PS5TrainingBridge
Void PS Box Example - 1 span, 9 girder system
07/25/16
36'-0"
24'-0"
6'-0"
6'-0"
Sidewalk Thickness 6"
Deck Thickness 6"
Sidewalk Thickness 6"
Travelway 1
G1 SII-48
G2 SII-48
G3 SII-48
G4 SII-48
G5 SII-48
G6 SII-48
G7 SII-48
G8 SII-48
G9 SII-48
2'-0"
8@4'-0" = 32'-0"
2'-0"